NAVAL POSTGRADUATE SCHOOL

MONTEREY, CALIFORNIA

THESIS

DEVELOPMENT OF A MEMS-SCALE TURBOMACHINERY BASED VACUUM PUMP

by

Michael J. Shea

June 2012

Thesis Advisor:	Anthony J. Gannon
Second Reader:	Garth V. Hobson

Approved for public release; distribution is unlimited

THIS PAGE INTENTIONALLY LEFT BLANK

REPORT DOCUMENTATION PAGE				*Form Approved OMB No. 0704-0188*

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instruction, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188) Washington DC 20503.

1. AGENCY USE ONLY *(Leave blank)*	**2. REPORT DATE** June 2012	**3. REPORT TYPE AND DATES COVERED** Master's Thesis

4. TITLE AND SUBTITLE Development of a MEMS-Scale Turbomachinery Based Vacuum Pump	**5. FUNDING NUMBERS**
6. AUTHOR(S) Michael J. Shea	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Naval Postgraduate School Monterey, CA 93943-5000	**8. PERFORMING ORGANIZATION REPORT NUMBER**
9. SPONSORING /MONITORING AGENCY NAME(S) AND ADDRESS(ES) DARPA	**10. SPONSORING/MONITORING AGENCY REPORT NUMBER**

11. SUPPLEMENTARY NOTES The views expressed in this thesis are those of the author and do not reflect the official policy or position of the Department of Defense or the U.S. Government. IRB Protocol number ______N/A______.

12a. DISTRIBUTION / AVAILABILITY STATEMENT	**12b. DISTRIBUTION CODE**
Approved for public release, distribution is unlimited	A

13. ABSTRACT (maximum 200 words)

This study forms part of a larger study to develop a MEMS scale turbomachinery based vacuum pump. This would allow very high vacuum to be drawn for handheld mass spectroscopy. This thesis concentrates on the roughing portion of the turbo pump where flow can still be treated as a continuum but the no slip boundary condition is not accurate. The first portion of this thesis investigates flow at Knudsen numbers ranging from 0.001 to 0.1. By using a first order analysis, the wall shear stress can be specified in a commercial computational fluid dynamics code allowing slip flow to occur. This method was validated against a basic Poiseuille flow at these higher Knudsen numbers where slip flow was present. This demonstrated that it was possible to use a commercial code to model Knudsen number flows between 0.001 to 0.1. The second part of the thesis focused on the design of a roughing pump stage consisting of three blade rows: a stationary inlet and outlet surrounding the rotor blade row. The no slip condition was not imposed as the simulated stage was assumed to be the outlet stage, and thus operating at a very low Knudsen number. A two dimensional analysis was developed to define the initial blade shape to achieve a maximum pressure ratio. A three dimensional simulation was developed to investigate the effects of tip leakage losses. The simulations are able to predict pressure ratio and power consumption of a particular stage of a MEMS scale turbopump. The final predicted pressure ratio of a stage with tip clearance is 1.0722 with power consumption of 0.4648 watts.

14. SUBJECT TERMS MEMS Turbomachinery, Computational Fluid Dynamics, Vacuum Pumps	**15. NUMBER OF PAGES** 115
	16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	**18. SECURITY CLASSIFICATION OF THIS PAGE**	**19. SECURITY CLASSIFICATION OF ABSTRACT**	**20. LIMITATION OF ABSTRACT**
Unclassified	Unclassified	Unclassified	UU

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. 239-18

THIS PAGE INTENTIONALLY LEFT BLANK

Approved for public release; distribution is unlimited

DEVELOPMENT OF A MEMS-SCALE TURBOMACHINERY BASED VACUUM PUMP

Michael J. Shea
Ensign, United States Navy
B.S., United States Naval Academy, 2011

Submitted in partial fulfillment of the
requirements for the degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

from the

NAVAL POSTGRADUATE SCHOOL
June 2012

Author: Michael J. Shea

Approved by: Anthony J. Gannon
Thesis Advisor

Garth V. Hobson
Second Reader

Knox Millsaps
Chair, Department of Mechanical and Aerospace Engineering

THIS PAGE INTENTIONALLY LEFT BLANK

ABSTRACT

This study forms part of a larger study to develop a MEMS scale turbomachinery based vacuum pump. This would allow very high vacuum to be drawn for handheld mass spectroscopy. This thesis concentrates on the roughing portion of the turbo pump where flow can still be treated as a continuum but the no slip boundary condition is not accurate. The first portion of this thesis investigates flow at Knudsen numbers ranging from 0.001 to 0.1. By using a first order analysis, the wall shear stress can be specified in a commercial computational fluid dynamics code allowing slip flow to occur. This method was validated against a basic Poiseuille flow at these higher Knudsen numbers where slip flow was present. This demonstrated that it was possible to use a commercial code to model Knudsen number flows between 0.001 to 0.1. The second part of the thesis focused on the design of a roughing pump stage consisting of three blade rows: a stationary inlet and outlet surrounding the rotor blade row. The no slip condition was not imposed as the simulated stage was assumed to be the outlet stage, and thus operating at a very low Knudsen number. A two dimensional analysis was developed to define the initial blade shape to achieve a maximum pressure ratio. A three dimensional simulation was developed to investigate the effects of tip leakage losses. The simulations are able to predict pressure ratio and power consumption of a particular stage of a MEMS scale turbopump. The final predicted pressure ratio of a stage with tip clearance is 1.0722 with power consumption of 0.4648 watts.

THIS PAGE INTENTIONALLY LEFT BLANK

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF ACRONYMS AND ABBREVIATIONS

Kn	Knudsen Number (-)
σ	Molecular Diameter (m)
k_B	Boltzmann Constant ($m^2 \cdot kg \cdot s^{-2}$)
υ	Kinematic Viscosity (m^2/s)
p	Total Pressure (Pa)
L	Characteristic Length (m)
ρ	Density (kg/m^3)
c	Speed of Light (m/s)
T	Temperature (K)
σ_v	Tangential Momentum accommodation coefficient (-)
u	Velocity (m/s)
τ	Shear Stress (Pa)
μ	Dynamic Viscosity (kg/(m·s))
λ	Mean Free Path (m)
$P_{o,inlet}$	Stagnation Pressure at the Inlet (Pa)
$P_{o,exit}$	Stagnation Pressure at the Outlet (Pa)
ANSYS	Multi Physics Simulations Software
CFX	Fluid modeling portion of ANSYS

THIS PAGE INTENTIONALLY LEFT BLANK

ACKNOWLEDGMENTS

First and foremost, I would like to thank the faculty and staff at the Naval Postgraduate School Turbo Propulsion Laboratory for not only assuming the capacity as the most influential instructors during my time at NPS but also for the guidance I received from them throughout my experience. Specifically, Dr. Anthony Gannon, who emphasized exceptional research and analysis techniques and providing advice when needed throughout the project. Also, Dr. Garth Hobson, who mentored and taught valuable engineering lessons both in the classroom and at the lab. John Gibson, for maintaining the facility to world-class standards and Chris Clay who devoted much of his time to developing the first computer aided designs for this project.

I would also like to thank CAPT Murray Snyder, USN for introducing me to the field of Computational Fluid Dynamics during my undergraduate studies at the US Naval Academy and providing me with adequate background and experience necessary to succeed in graduate school.

Outside of Academia, I would like to thank the family, friends and loved ones who have supported me throughout my entire educational career and will continue to support me in years to come.

Lastly, I would like to thank the Defense Advanced Research Projects Laboratory, DARPA, under the direction of Dr. Tayo Akinwande and Dr. Michael Wolfson for providing funding for the project.

THIS PAGE INTENTIONALLY LEFT BLANK

I. INTRODUCTION

An increasing demand in both the private and military sectors for small machinery carries with it a necessity to further understand the engineering fundamentals behind these micro devices. However, as the field of material science advances with the use of photolithography for microfabrication, machinery is continually able to decrease in size. The smaller turbomachinery demands a further understanding of the physical fundamentals of small scale flows. The microchannel is a key component of micro-electro-mechanical-systems (MEMS) as they are the used to transport fluid in such systems.

It has been known for quite some time that the basic engineering assumptions which were previously accepted at the macro scale are no longer valid at the micro scale. This history of flows in microchannels can be traced back as early as Knudsen in 1909 [5]. It recently reappeared on the agenda of fluid researches in the early 1990s with work by Pfahler and Beskok. [9]

The field of fluid mechanics primary exists in the continuum regime where the Navier-Stokes, conservation of mass and conservation of energy can be solved simultaneously to analyze the flow in a macro-scale computational model. At extremely low vacuums and very small geometries the flows can be treated on a molecular level and solved through the use of the Boltzmann Equations. At slightly higher pressures and larger geometries the flow can be treated as a continuum but the no-slip boundary condition no longer applies. This is brought about by the kinetic theory of gas and the Boltzmann equation, the study of fluid on a micro and nano level must be approached differently. Instead of the no slip assumption, the dilute gas assumption is made and applied to the Chapman-Enskog theory which allows for modeling of gases at smaller scales than normally acceptable with a classical Navier-Stokes approach. [6]

Experimentally, flow phenomena have been studied at these small geometries that are not seen at larger scales. For instance, as measured by Pong, Ho & Liu in 1994 [8] and by Arkilic, Schmidt & Breuer in 2001, the pressure distribution is not linear. [4] The goal of this research is to computationally study the slip regime in an effort to apply it to the design of a micro-scale vacuum pump.

A. SLIP FLOW BACKGROUND

As mentioned, with most applications engineers have seen, the assumption that fluid is continuous can be made and this is the basis of continuum mechanics. Here, both the continuity, Navier-Stokes and energy equations are valid with the assumption of a no-slip boundary condition. This assumption allows for easy analytical solutions to basic cases of fluid flow such as Poiseuille and Couette flow. Since the velocity gradient can be calculated from the known velocities at the wall, it became elementary to calculate shear stresses and drag forces. As the pressure of a flow drops and the passage geometry decreases the wall velocity cannot be assumed to be zero.

The non-dimensional number which describes this regime is the Knudsen number. This is defined as follows:

$$Kn = \frac{\lambda}{L} \tag{1}$$

This ratio between the mean free path and the characteristic length can range from infinitesimally small to numbers greater than ten. The no slip regime is assumed when this number is less than 0.001.

In cases pertaining to the present study, the Knudsen Number ranges between 0.001 and 0.1. Here the flow can still be treated as a continuum but the no-slip assumption is no longer accurate. The number is brought up by either decreasing the characteristic length or increasing the mean free path of the fluid by changing the pressure and/or the temperature. The mean free path is:

$$\lambda = \frac{1.051 k_B T}{\sqrt{2}\pi {\sigma_v}^2 p} \tag{2}$$

In general, at standard atmospheric temperature and pressure, the mean free path is about 65-70 nm. When the Knudsen number reaches the aforementioned higher order of magnitudes, the assumption that the velocity of the wall is equal to the velocity of the fluid can no longer be made and this must then be taken into account when modeling such devices.

It can be proven that based on the Maxwellian slip boundary condition the slip velocity as a function of the velocity gradient near the wall is [2]:

$$u_{wall} = \frac{2-\sigma_v}{\sigma_v} Kn \left. \partial u \middle/ \partial y \right|_{wall} \tag{3}$$

Beyond this point, as the Knudsen number becomes increasingly larger, the flow enters the transition regime where its Knudsen number is between 0.1 and 10. Here the Navier-Stokes equations are not valid as the flow ceases to behave as a continuum and the Boltzmann equations must be used to solve this flow. Here, each molecule is treated as a different element colliding with other molecules. Taking the Knudsen number beyond 10, all molecular collisions are neglected in the Boltzmann equations because in this region, the mean free path is 10 times larger than the channel where the gas is traveling, thus most likely not colliding. The scope of work studied in this paper is limited to Knudsen numbers between 0.001 and 0.1 placing the fluid in the slip and continuum regimes.

The study begins with modified Poiseuille flow. The aim of this was to investigate whether a commercial code could be modified to model slip flow. At these very small pressures, the mean free path of the flow will increase thereby increasing the Knudsen number and bringing the flow into the slip regime. Per initial design, this centrifugal “chip pump” will operate at 200,000 revolutions per minute with 5um tip gaps. The ultimate goal of this study is to apply this technique to a MEMS style vacuum pump, although the latter was not completed in this study.

B. VACUUM PUMP BACKGROUND

Vacuum pump technology has existed since the middle of the 20th century. Modern vacuum pumps operate in two parts. A roughing pump performs the initial vacuum setup. This component is what is being investigated in this study. The name comes from the fact that the pump “roughs” the majority of the system and provides some backing for the diffusion pump. This is because the diffusion process cannot function while diffusing into atmospheric conditions. These back-to-back compressors combined with a complex valve structure create the basis for a vacuum pump. On a large scale, this is practical since the diffusion pump is designed to deal with extremely low pressure and the roughing pump is of a large magnitude. In the case of the mass spectrometer, shown in Figure 1 below, the large roughing pump can be seen on the left hand side.

Figure 1. Mass Spectrometer (After: [1])

The difficulty of simulating such a device occurs when the roughing pump becomes very small as the flow enters the slip regime. It is understood the suction side of the roughing pump will exhibit slip flow characteristics, and this is effective at a large scale with today's technology. However, translating this to a MEMS scale that can operate at atmospheric conditions on one end and light vacuum conditions on the other drives the motivation to model and understand the slip flow.

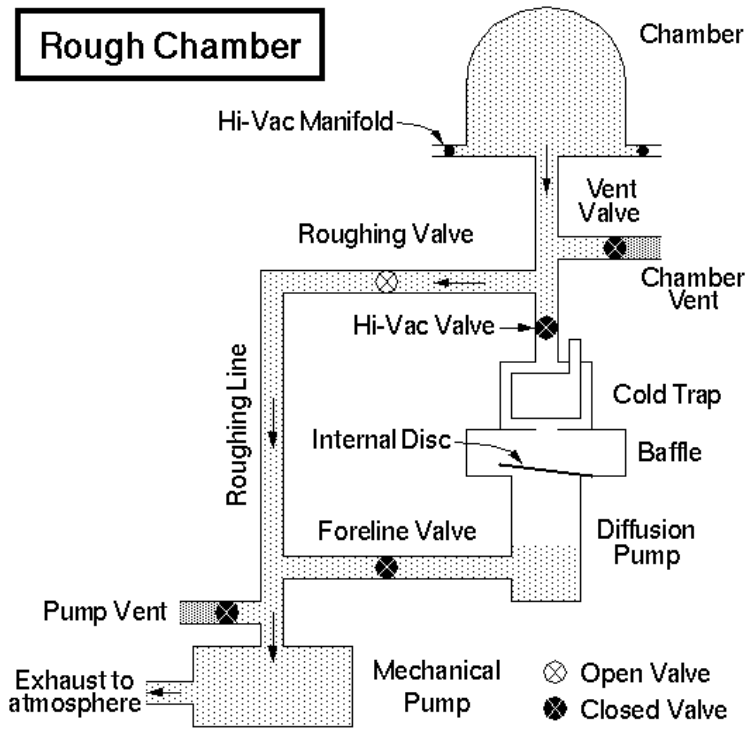

Figure 2. Description of a Basic Vacuum System (From: [10])

Figure 2 is a visual representation of how a typical vacuum pump works and is the basis for a MEMS scale pump.

C. MASS SPECTROSCOPY BACKGROUND

The ability to create miniature vacuum pumps has various applications, primarily in the capability of handheld mass spectroscopy. A brief understanding of how these devices work will help to better appreciate the need to study slip flow at this very small scale. Mass spectroscopy has countless battlefield applications from a military perspective in the detection of Chemical, Biological and Radiation (CBR) elements which can be used as weapons. In today's environment, this is only possible by collecting samples and bringing them to a lab where the mass spectrometers are located. Designing devices that can be carried by a human or unmanned vehicle, mitigates the time delay associated with CBR detection in the field and can transmit real-time data for analysis and operational decision making.

The concept of a mass spectrometer is that whichever substance is being studied must be separated into individual atoms. For detecting particles in air, this must be accomplished by bringing ambient air condition air into a high vacuum. This reason is the major setback for handheld mass spectrometers and therefore advances in the understanding of fluids at such vacuum pressures and high speeds are necessary. The vacuum pressures are necessary to prevent the molecules from colliding with each other during the testing and analysis phase.

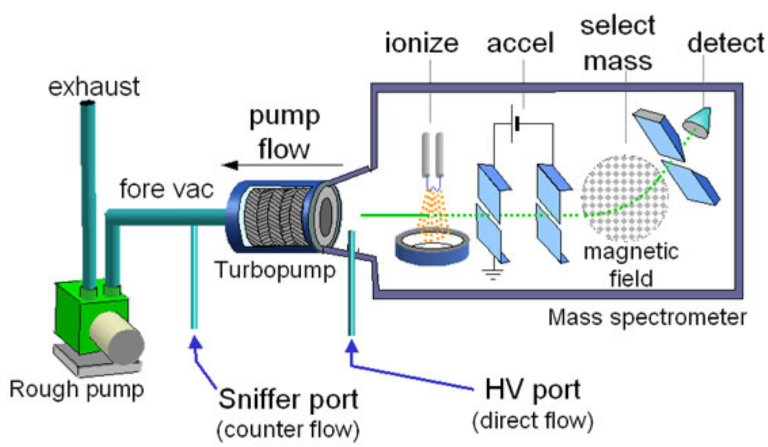

Figure 3. Mass Spectrometer diagram (From: [1])

It works by individual atoms being ionized by having electrons fired at them. These ions are then subject to various electric and magnetic fields. The displacement of theses ions as they travel through the magnetic field can accurately provide details about the specific atom traveling through the machine. As shown in Figure 3, in order for this test to work vacuum pressures must be maintained throughout the test section. This allows for the ions being measured to not be affected by anything else except the magnetic field. Typically, the vacuum drives the size of the entire system and by developing such types of MEMS turbopumps, handheld CBR detection through mass spectroscopy will be possible.

D. PROCEDURE

The study begins with an analysis of slip flows using a basic Poiseuille analysis. A commercial code CFX was modified to include slip flow conditions. Following this transient two-dimensional and three-dimensional designs of a typical MEMS scale turbopump stage were simulated. In the time available it was not possible to include the no-slip conditions in these models.

THIS PAGE INTENTIONALLY LEFT BLANK

II. SLIP FLOW VALIDATION

A. SOLUTIONS TO SLIP FLOW EQUATIONS

1. Shear Stress at the Wall

It has been shown [6] that the general solution for the velocity at the wall can be estimated using Equation 4:

$$u_{gas} - u_{wall} = \frac{2-\sigma_v}{\sigma_v} Kn \left.\frac{\partial u}{\partial y}\right|_{wall} \tag{4}$$

As the Knudsen number approaches zero, which is a safe assumption for most flows in the continuum regime, the velocity at the wall also approaches zero. Using Newton's law of viscosity for the shear stress,

$$\tau = \mu \left.\frac{\partial u}{\partial y}\right|_{wall} \tag{5}$$

Equation 4 can be rearranged as:

$$\frac{u_{gas} - u_{wall}}{\frac{2-\sigma_v}{\sigma_v} Kn} = \left.\frac{\partial u}{\partial y}\right|_{wall} \tag{6}$$

to obtain an expression for the shear stress, τ,

$$\tau = \frac{\left(u_{gas} - u_{wall}\right)\mu}{\frac{2-\sigma_v}{\sigma_v} Kn} \tag{7}$$

This is an important step in developing an appropriate equation for the shear stress at the wall. The assumption of a very rough wall, or $\sigma_v=1$ is a good approximation for micro-channel flow because of the relative roughness of the wall to the mean free path of the air traveling through. This can be further simplified for the test case of plate Poiseuille flow, a pressure driven flow in a cavity such as seen in a typical water pipe.

$$\tau = \frac{u_{gas}\mu}{Kn} \tag{8}$$

This expression, used in conjunction with a calculated Knudsen number can determine shear stress. The calculation for Knudsen number tabulated at various pressures for the test geometry can be found in Appendix A.

2. CFD Validation Equation

Once the shear stress is specified, the average and maximum velocities for both the slip and no slip cases can be calculated using CFX Solver. The setup which will be described was run with both the slip and no-slip cases. The results were used to find the necessary values for calculation purposes. For a generic no-slip Poiseuille flow, it has been shown that the maximum velocity is

$$u_{no\ slip} = \frac{Re}{2}\frac{dP}{dx}\left(y^2 - y\right) \tag{9}$$

Due to the parabolic nature of the fluid flow, the maximum velocity is simply Equation 5 evaluated at y=0.5. Therefore, the maximum velocity for a no slip case is:

$$u_{max\,no\ slip} = \frac{Re_{max\,no\ slip}}{2}\frac{dP}{dx}\left(\frac{-1}{4}\right) \tag{10}$$

For the slip case, the parabola is shifted downstream which signifies a measurable velocity at the wall. To account for this shift the previous two equations can be rewritten for the slip case.

$$u_{slip} = \frac{\mathrm{Re}}{2}\frac{dP}{dx}\left(y^2 - y - \alpha\right) \tag{11}$$

$$u_{\max slip} = \frac{\mathrm{Re}}{2}\frac{dP}{dx}\left(\frac{-1}{4} - \alpha\right) \tag{12}$$

To get the average velocities, the equations for slip and no slip velocity are simply one half of the maximum velocities. Therefore, putting the following terms together, the following equation can be produced to validate four computable quantities:

$$\left(\frac{u_{max\,no\,slip}}{u_{max\,slip}}\right)\left(\frac{u_{mean\,slip}}{u_{mean\,no\,slip}}\right)=\frac{\frac{Re_{max\,no\,slip}}{2}\frac{dP}{dx}\left(\frac{-1}{4}\right)}{\frac{Re_{max\,no\,slip}}{2}\frac{dP}{dx}\left(\frac{-1}{4}-\alpha\right)}\left(\frac{u_{mean\,slip}}{u_{mean\,no\,slip}}\right)$$

$$=\frac{u_{mean\,no\,slip}\left(\frac{-1}{4}\right)}{u_{mean\,slip}\left(\frac{-1}{4}-\alpha\right)}\left(\frac{u_{mean\,slip}}{u_{mean\,no\,slip}}\right)=\frac{1}{1+4\alpha} \tag{13}$$

Rearranged, the following can be established:

$$4\alpha=\frac{u_{max\,slip}u_{mean\,no\,slip}}{u_{max\,no\,slip}\;u_{mean\,slip}}-1 \tag{14}$$

Where

$$\alpha=\frac{2-\sigma_v}{\sigma_v}Kn \tag{15}$$

This is a useful step because it is now possible to use this equation to allow validation of the computed velocities against the Knudsen number. Again, this is a result of the assumption that $\sigma_v=1$.

$$4Kn=\frac{u_{max\,slip}u_{mean\,no\,slip}}{u_{max\,no\,slip}\;u_{mean\,slip}}-1 \tag{16}$$

If the first order assumption is correct, there should be a linear relationship between the Knudsen number and the calculated mean and maximum velocities for no-slip and slip flow.

B. COMPUTATIONAL SETUP

1. Geometry

For the Poiseuille flow validation case, the geometry used was a basic rectangular channel as shown in Figure 4.

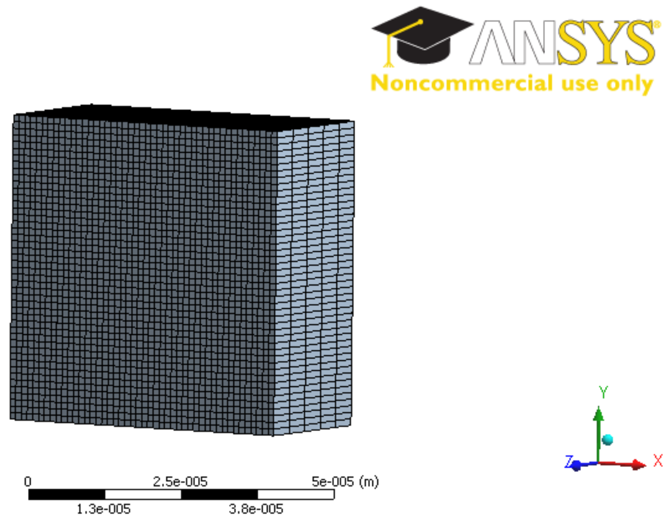

Figure 4. Geometry for Poiseuille Validation

The dimensions as shown are 50 microns by 25 microns. This was designed to be half of a symmetric channel cross section for ease of calculation purposes. A simple geometry like this is not very computationally expensive; therefore, meshing is not as crucial as it will be in later simulations. A mesh refinement study was conducted and had no effect on the results of this validation.

2. Meshing

The meshing was simple because of the basic geometry involved in this simulation. A sweep method is able to be used with five cells across due to the fact that the outer walls of the channel exhibit a symmetry boundary condition. Because CFX is inherently a three-dimensional solver, it will not run with only one element across. That will create a of circular reference and not produce logical results, if any are produced at all. There were ultimately 10,580 elements in the mesh.

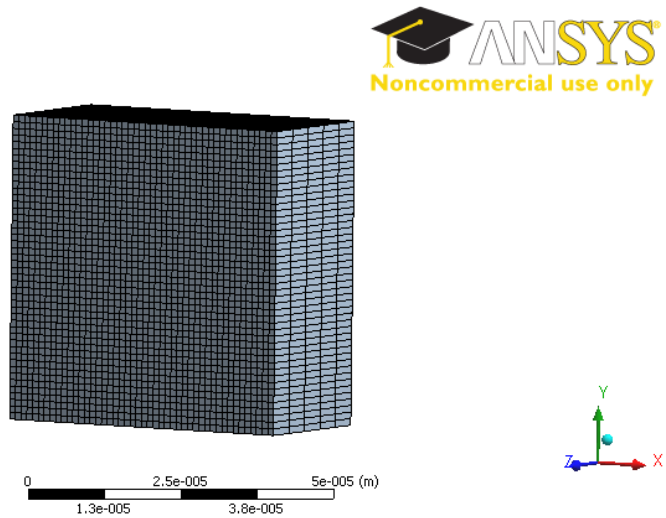

Figure 5. Poiseuille Validation Mesh

3. Setup

Correctly setting up the boundary conditions is the most important part of this validation process. There were two different setups for the different scenarios, one including slip flow and the other making the no-slip assumption. In this particular case, since the passage is so small, driving the Reynolds number lower, laminar flow is assumed.

a. Domain

Throughout this entire process the fluid used was assumed to be air as an ideal gas. The reference pressure, which drives the Knudsen number, can be modified to collect different validation test points at different locations.

b. Boundary Conditions

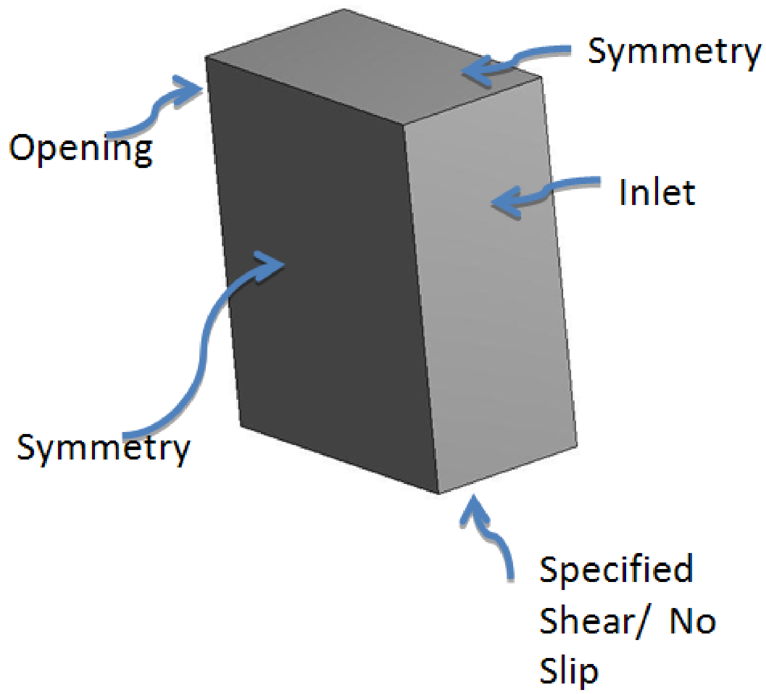

Figure 6. Description of Boundary Conditions for Validation

In both cases the top wall in addition to the two sides were set up as symmetry walls. The flow was driven through the section by a pressure differential between the inlet and the opening. The pressure drop across this 50 micron length was 2 Pascals. Although 2 Pascals seems insignificant, the pressure drop per unit length was:

$$\frac{dP}{dx} = \frac{2Pa}{50 \times 10^{-6} m} = 40{,}000 \, {}^{Pa}\!/_{m} \tag{17}$$

The low pressure, varying from 200 Pascals to 2,000Pascals, in addition to the small characteristic length of the geometry, provided for the necessary Knudsen number to induce slip flow.

c. Slip and No Slip Conditions

CFX has four built in functions when setting up a wall boundary condition. The four built-in options are as follows: no slip, free slip, finite slip and specified shear. The no slip condition works for the continuum regime where the shear stress is equal to the product of the velocity gradient at the wall and the viscosity of the fluid. However, when attempting to represent the slip regime, an expression must be written to describe the shear stress at the wall. This expression is the same that was mentioned before as:

$$\tau = \frac{u_{gas}\mu}{Kn} \tag{18}$$

The setup can be viewed in more detail in Appendix B.

C. RESULTS

After successful runs of this simple geometry under the aforementioned setup, the centerline velocity as well as the average velocity were able to be calculated. The centerline velocity was simply the maximum velocity and the average velocity was calculated using the following expression:

$$\bar{v} = \frac{\dot{m}}{\bar{\bar{\rho A}}} \tag{19}$$

Using this, the average and maximum velocities were tabulated while varying the reference pressure from 200Pa to 2,000Pa. This allowed for the calculation of:

$$4Kn = \frac{u_{max\ slip}u_{mean\ no\ slip}}{u_{max\ no\ slip}\ u_{mean\ slip}} - 1 \tag{20}$$

Figure 7 shows the computed velocity profile at a relative pressure of 200Pa. The slip velocity follows the same parabolic shape, just translated to account for the slip velocity. This is characteristic of a first order approximation of slip velocity. The tabulated results at different Knudsen numbers can be found in the table located in Appendix C and plotted in Figure 8, the Knudsen number versus (21).

$$1/4\left(\frac{u_{max\ slip}u_{mean\ no\ slip}}{u_{max\ no\ slip}\ u_{mean\ slip}} - 1\right) \tag{21}$$

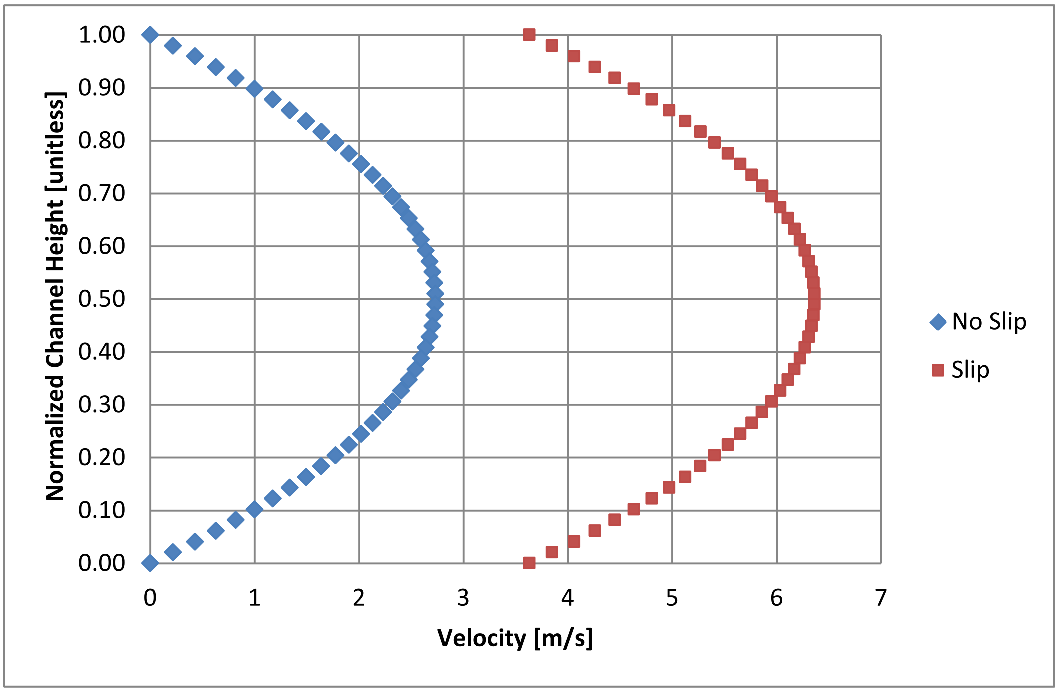

Figure 7. Velocity Distributions of Slip and No-Slip Flow at 200Pa Relative Pressure

D. DISCUSSION

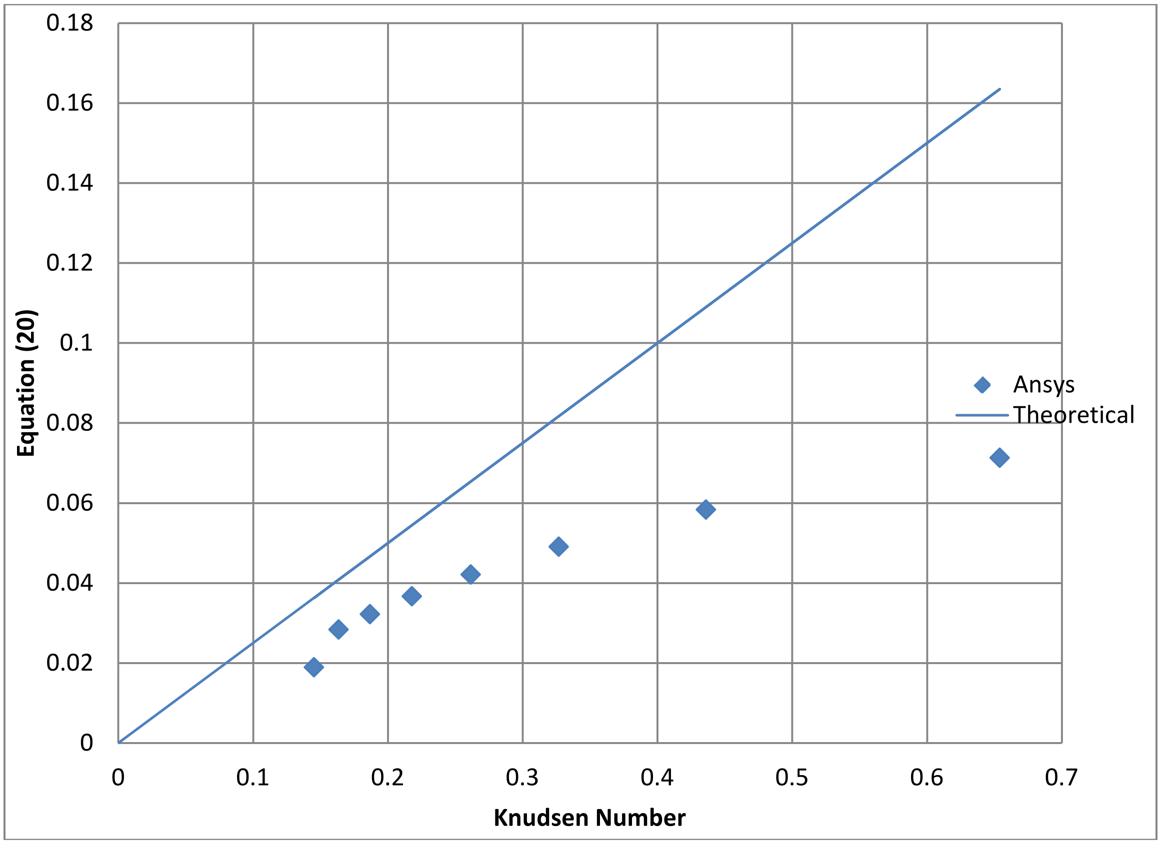

Figure 8. Plot of Slip Flow Validation

As shown in Figure 7 and Figure 8, the expression for the shear stress while under the slip condition definitely calculated a slip velocity at the wall as opposed to the zero velocity condition in the no slip case. However, the first order approximation which predicts the relationship between the maximum and average velocities of the slip and no slip boundary conditions does not match up perfectly with what CFX calculated. In future studies, a second degree analysis of the shear stress in the slip flow regime should be considered for comparison purposes. This will allow not only the parabolic profile to be shifted downstream, but also allow for a non-parabolic flow distribution. The general trend, however, of the slip flow exhibiting larger maximum and average velocities holds thus proving the effectiveness of the expression.

THIS PAGE INTENTIONALLY LEFT BLANK

III. TWO-DIMENSIONAL SIMULATIONS OF MICRO TURBOPUMP

The design of the initial micro turbopump was driven by the geometric constraints of the vacuum pump manufacturing process. This resulted in a planar centrifugal-style pump with concentric rotor and stator stages. Photolythography, the micro-etching process which will be used, limits designs to flat surfaces. For ease of calculation and to improve computational efficiency, the entire model was constructed using SolidEdge but the actual computational molecule consisted of two rows of stator blades and one row of rotor blade. For the two-dimensional case, these blades were modeled with symmetry conditions at both ends thereby simulating infinite span.

A. FIRST DESIGN ITERATION

1. Geometry

The ability for Solid Edge to create interrelated variables for updating design features of this pump made it a desirable solid modeler for the project. This allowed the compressor blades to be modified without actually having to create new drawings and repattern the blades around the disk. The drawings of existing designs were obtained and modeled using SolidEdge. To accurately model the initial designs, the blade counts of each individual stage were tabulated and the shape of each blade was measured. The initial design's first stage consisted of three blade rows: two stationary stator rows with the rotor blade sliding between them. There were 696 and 684 stator blades in the outer and inner row respectively and 68 rotor blades. Each dimension was set as a variable in solid edge to allow modification of the shape and count of each blade. The rotor and stator were created as two separate parts and combined in an assembly. Although only one stage was simulated for computational efficiency, the actual machine will consist of 100 stages.

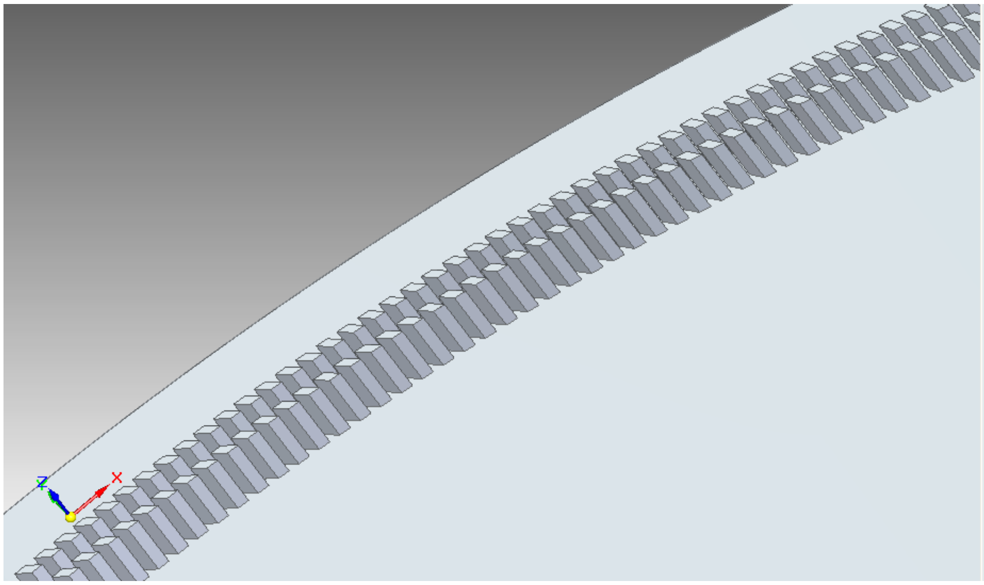

Figure 9. Initial Stator Design

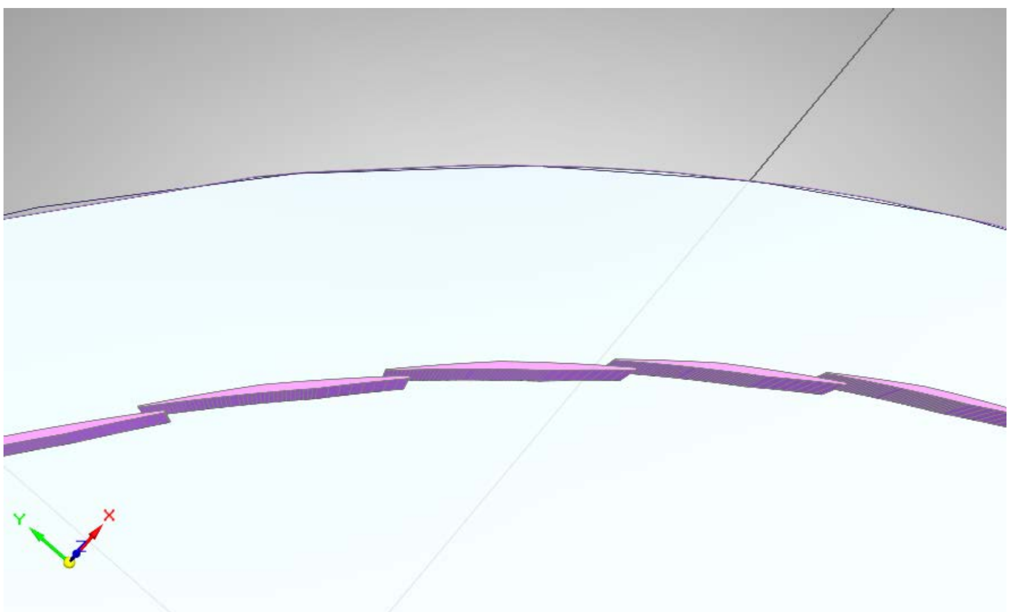

Figure 10. Initial Rotor Design

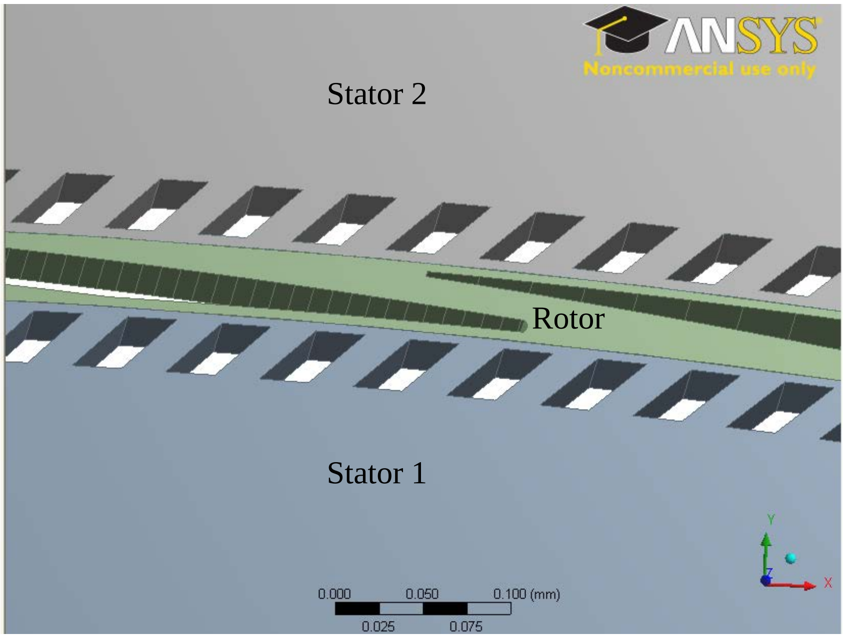

Figure 11. Gas Path Model in ANSYS/CFX of Initial Two-Dimensional Design

The assembled rotor and stator were used to cut the gas paths for importing into CFX. Three gas paths, one for each stage were imported into CFX. The initial study was two-dimensional and so symmetry conditions were used on the top and bottom surfaces.

2. Meshing

The mesh of 64,560 million elements was generated using a swept mesh method in the span-wise direction while a fine refinement was set, providing a good two-dimensional mesh. Figure 12 shows the coarseness of the mesh away from the blades area. Figure 13 shows the fine mesh in between the blade stages. A match control was also used for all of the rotational simulations done in both the two-dimensional and three-dimensional realm. This means that the elements on one side of the sector match up perfectly with the elements on the other side of the sector. This is necessary to save computational time while properly simulating only a small sector of the pump.

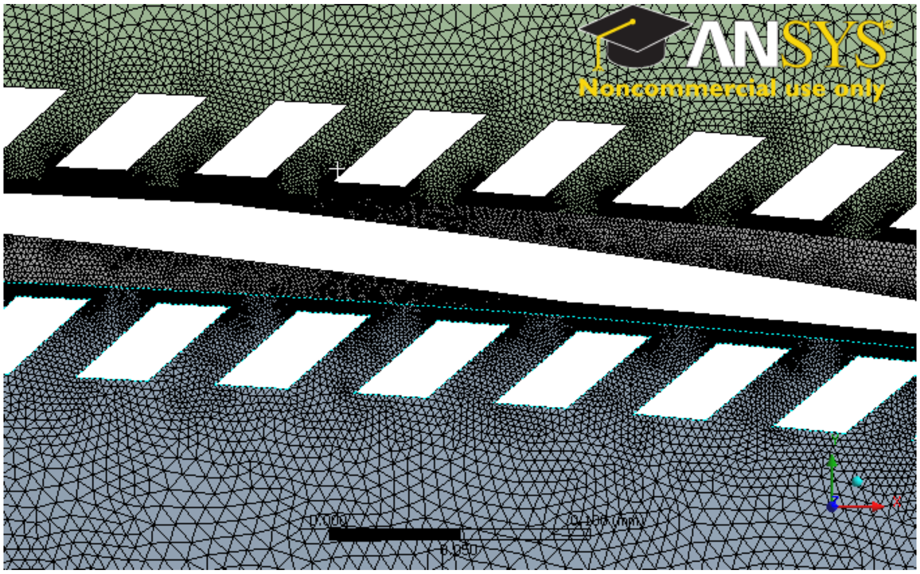

Figure 12. Mesh for Two-Dimensional Iteration 1

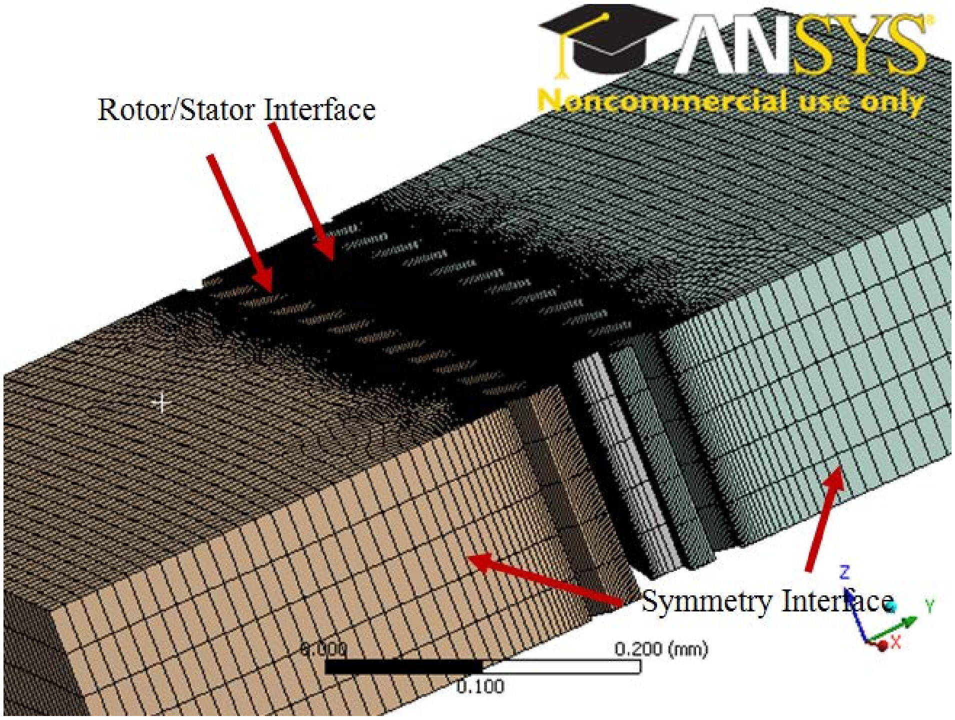

Figure 13. Swept Method of Mesh for Two-Dimensional Iteration 1

3. Setup

The setup for this geometry assumes a two dimensional turbopump. This is accomplished by assigning symmetry conditions at the top and bottom of the model. In addition, periodic boundaries were set on the radial boundaries, simulating the full rotor and stator rows. The detailed setup of the simulation can be found in Appendix D. The setup of this rotor-stator assembly depends greatly on the interfaces. The radial faces have a symmetry interface, annotated in Figure 13, simulating a complete stage while being computationally more efficient. There are also moving interfaces between the rotor blades and the stator blades. The transient and steady state runs have very similar setups, with a few modifications. The complete steady-state setup is presented in Appendix D while the transient is presented in Appendix E.

4. Results

The simulation run for this with 100 iterations of assumed steady state flow assuming averaging of the flow across the moving interfaces The transient simulations were then run until the pressure and power began to converge to a consistent sinusoidal motion. Figure 14 shows the convergence of the pressure ratio minus one to an average value of approximately 2.71E-02. The values of pressure ratio were normalized to 1 for ease of reporting minor changes. However, realistically, the pressure ratio is about 1.0271. With its very small scale and the fact that the pump was working in a constant effort to drive the air out of the center of the pump, the convergence required of the observed parameters (power and pressure ratio required a high number of iterations. this seems computationally expensive, the lack of a turbulence model as well as the two dimensional nature of the problem meant it ran relatively quickly using x parallel processes on an x MHz machines. Typical two-dimensional runs took x hours. This flowed inward, creating a vacuum on the outer portion of the stage.

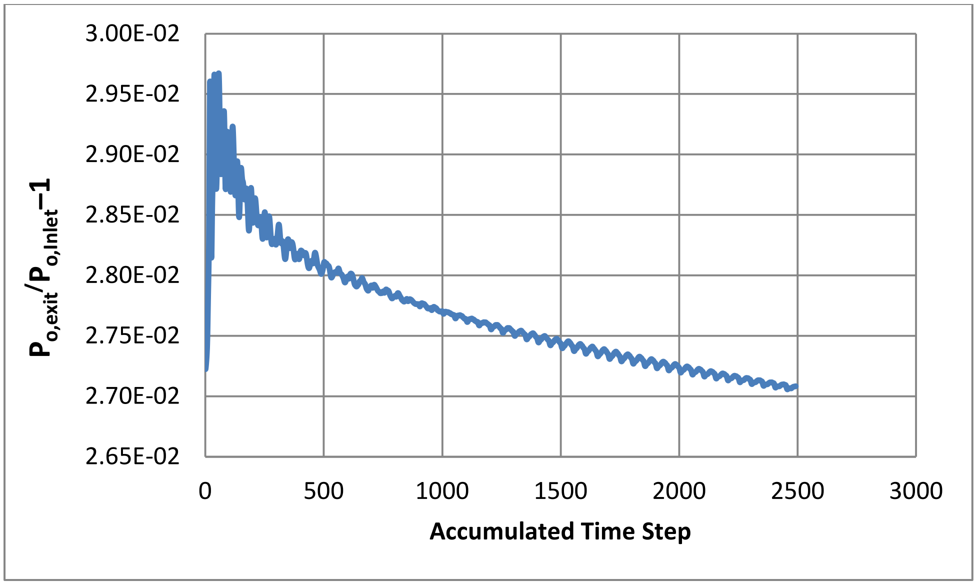

Figure 14. Pressure Ratio Two-Dimensional First Design Iteration

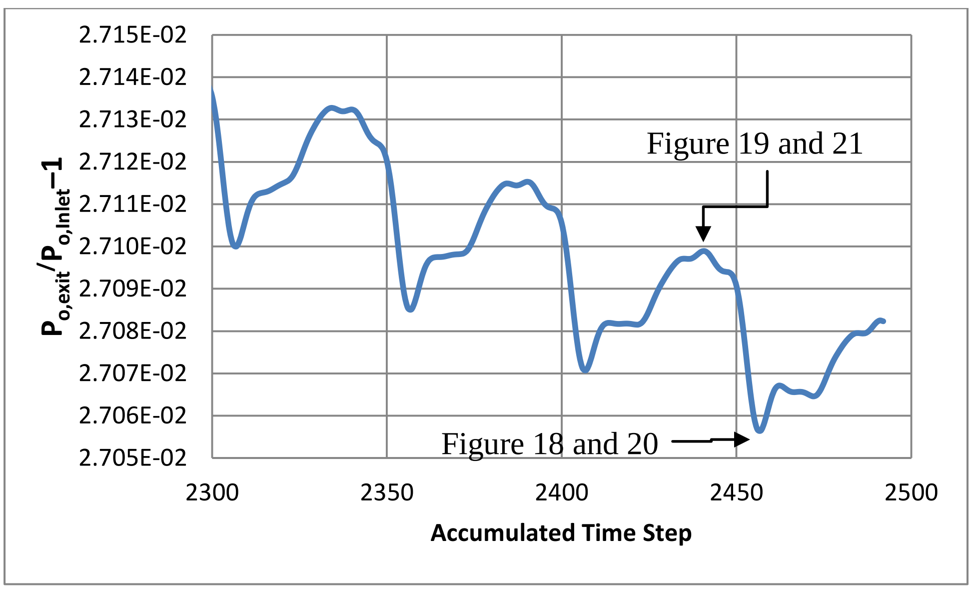

Figure 15. Zoomed in Pressure Ratio Two-Dimensional First Design Iteration

Looking at the other monitored point throughout the simulation, the blade power, it can also be seen in Figure 16 and Figure 17 that it operates at a very low power, oscillating with the pressure ratio converging to a value of 1.9E-7 J.

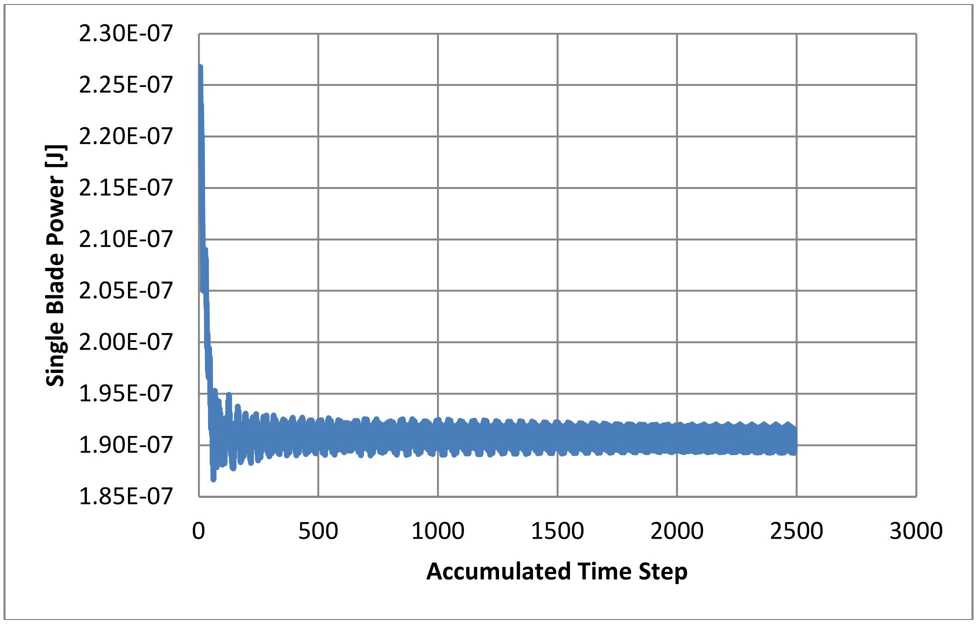

Figure 16. Single Blade Power for Two-Dimensional First Design Iteration

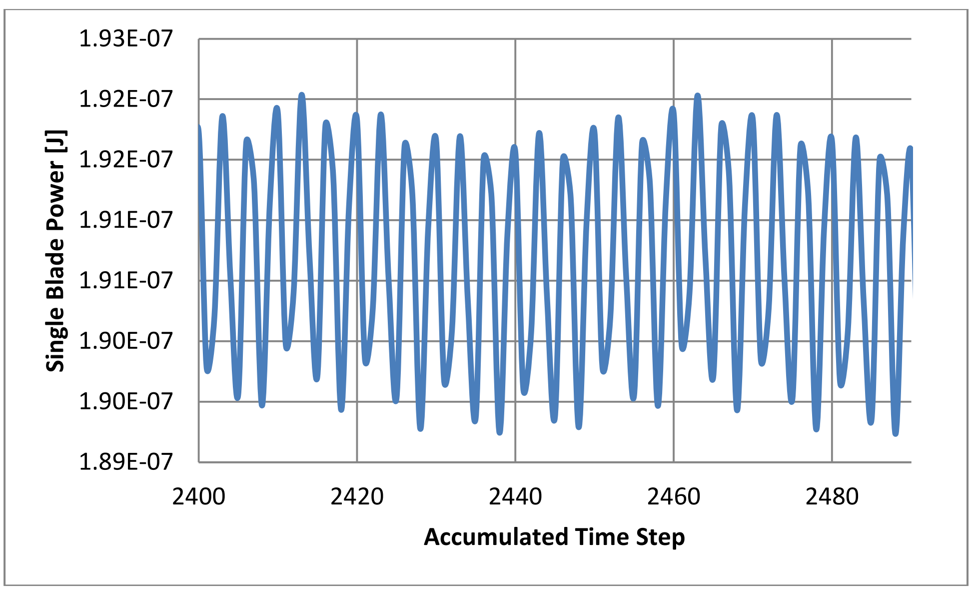

Figure 17. Zoomed In Single Blade Power for Two-Dimensional First Design Iteration

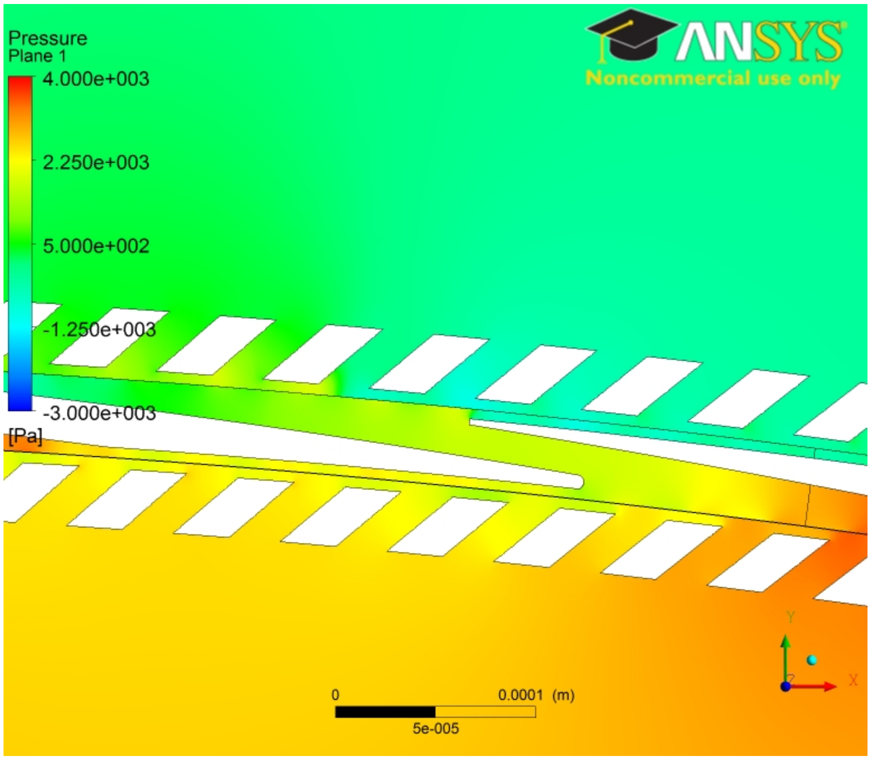

Figure 18. Pressure Distribution at Minimum Pressure Ratio

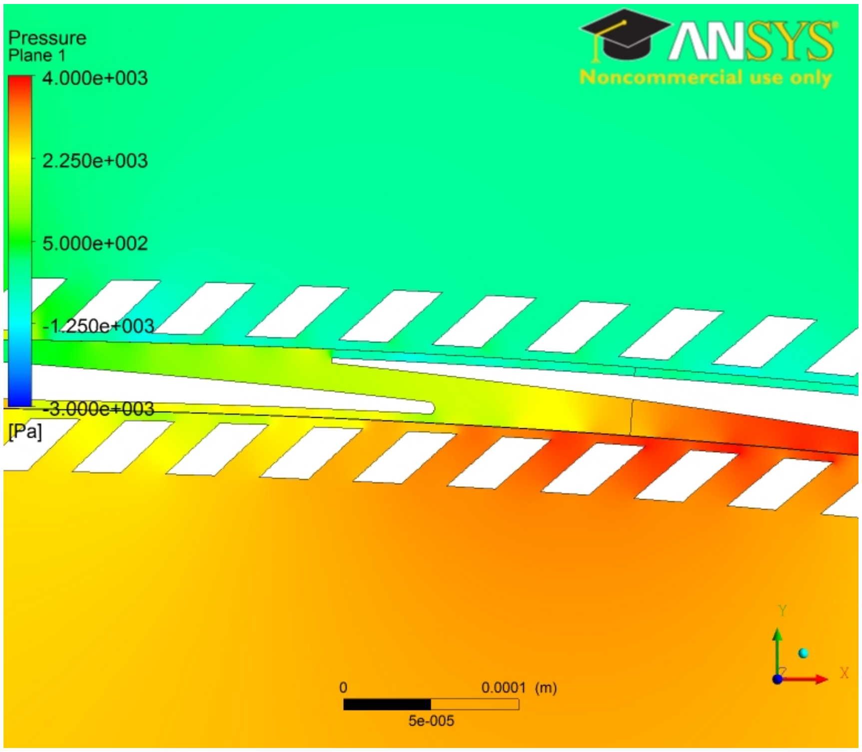

Figure 19. Pressure Distribution at Maximum Pressure Ratio

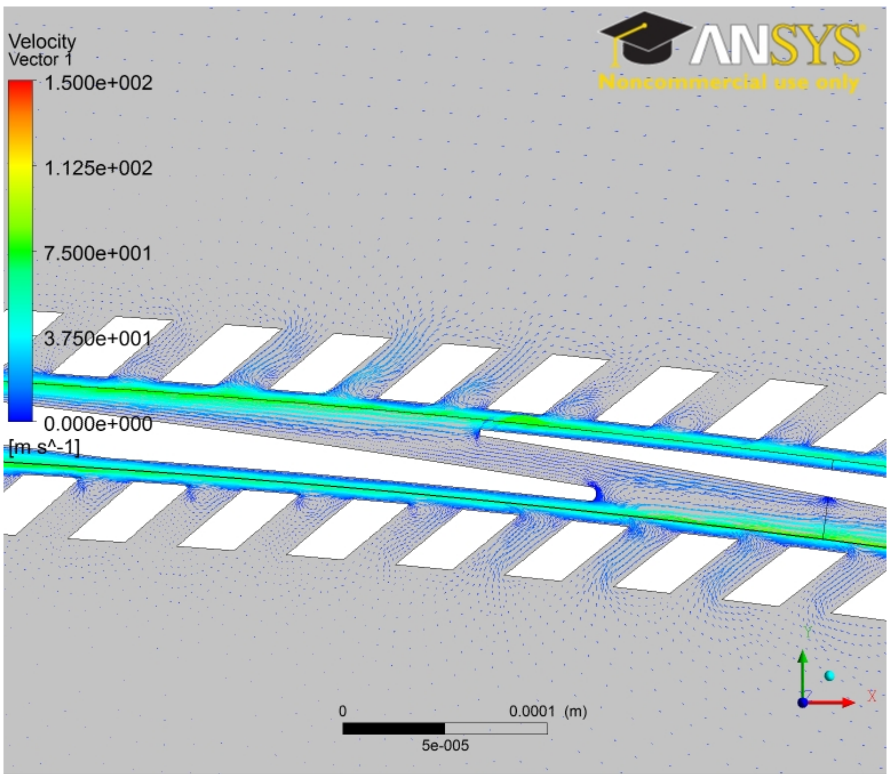

Figure 20. Velocity Vector Field at Minimum Pressure Ratio

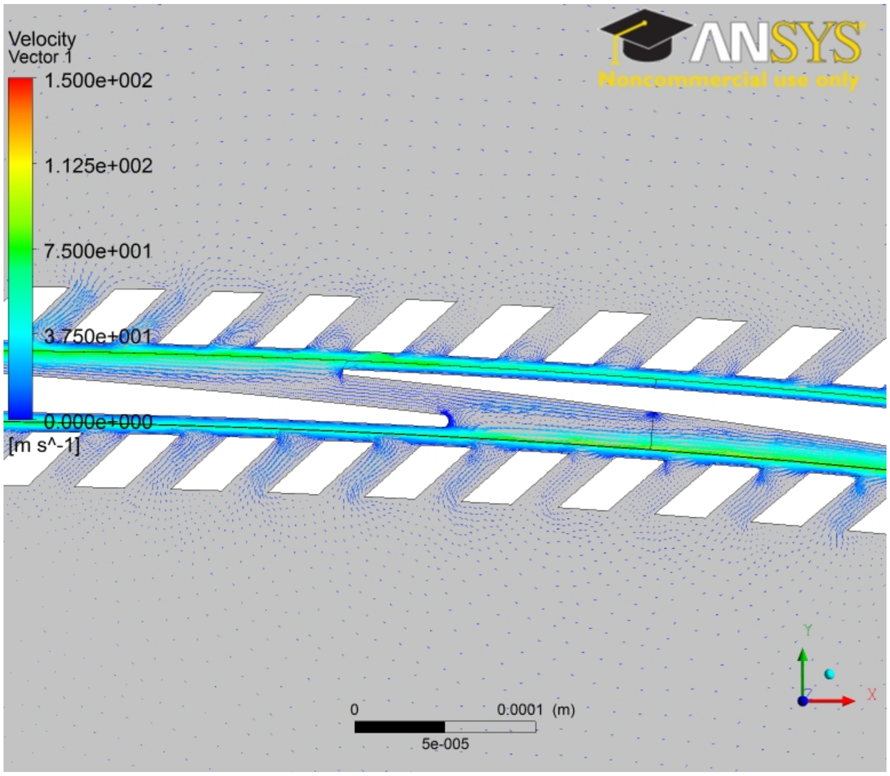

Figure 21. Velocity Vector Field at Maximum Pressure Ratio

The vector field at the maximum pressure ratio, Figure 21, shows a minimization of flow escaping from the vacuum. This design drives the air towards the center of the pump, thereby creating a vacuum pressure on the outside and a higher pressure on the inside. Figure 18 and Figure 19 both further emphasize this. In Figure 19, the maximum pressure ratio case, the rotor blade has the very high pressure region in between itself and the high pressure stator.

B. SECOND DESIGN ITERATION

1. Geometry

The second design iteration consisted of a more traditional centrifugal concentric stage pump where the flow is pumped from the inner radius to the outer radius. Although the procedure for creating these is similar to that of the first iteration, the blades themselves are slightly different. These blades have more camber, a longer chord and additional overlap. Different length blades were simulated to try and maximize the pressure ratio of the stage. Figure 22 shows the 34 Blade count with a 1500 micron chord on the same 16 millimeter diameter disk used in the first iteration in II.B.i.

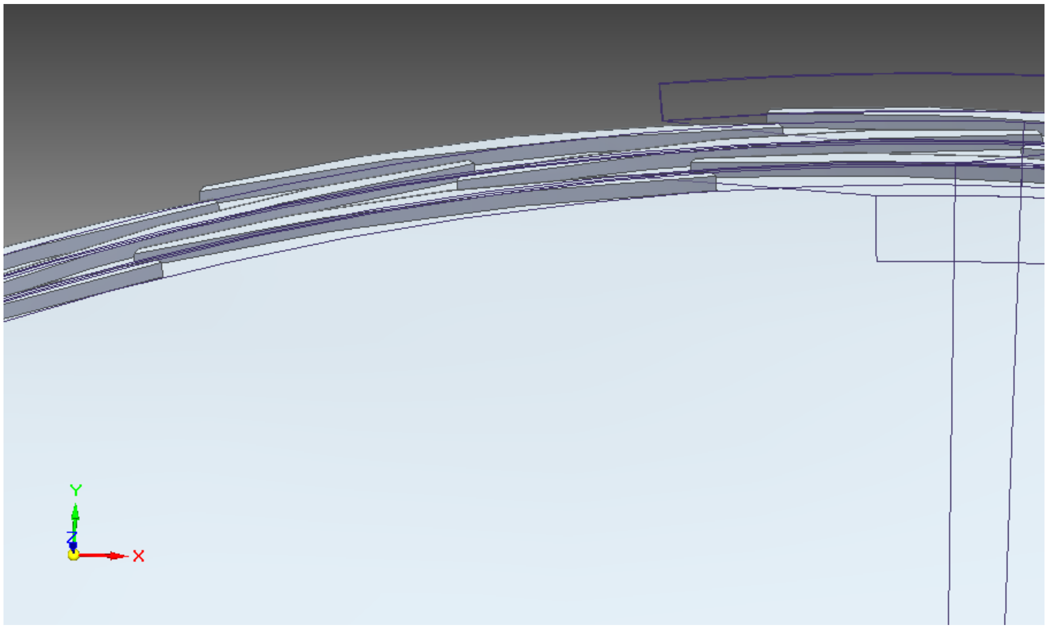

Figure 22. Geometry for 2nd Iteration of Two-Dimensional Simulation

2. Meshing

The mesh settings were similar to that of the initial design. Just as in the initial validation, it was not necessary to use many elements in the span-wise direction due to the two-dimensional nature of the problem and the symmetry conditions which accompany it. However, it is necessary to refine the mesh along the leading and trailing edges of each blade to more accurately model the air as an ideal gas through the very low pressures. Figure 23 shows the mesh of the air molecules with 111,565 elements.

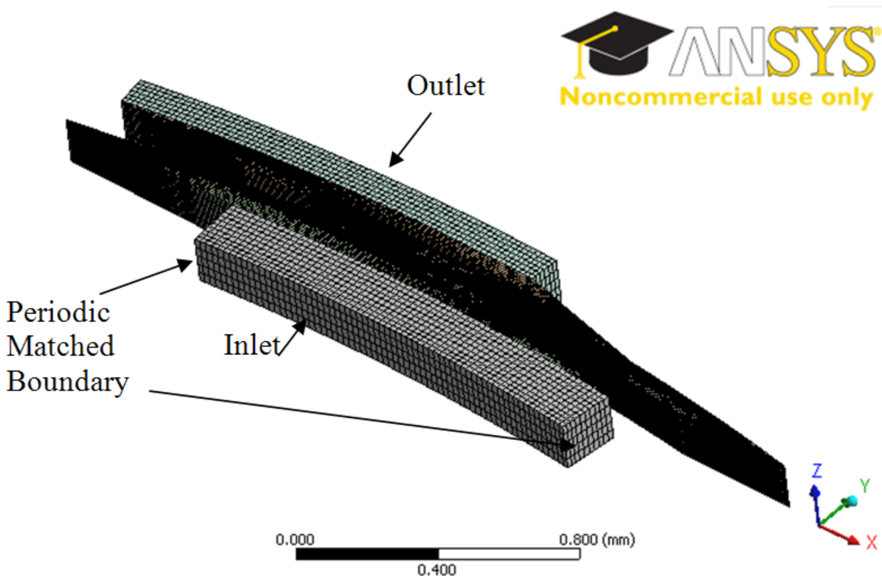

Figure 23. Mesh of Second Two-Dimensional Iteration

3. Setup

Similar to the meshing and initial geometry, the setup was very similar to that of the initial design. The top and bottom faces were set up as symmetry conditions whereas edges of each stage were set up as matched periodic boundaries as shown in Figure 23. As before a steady-state solution was needed to provide initial conditions to the transient simulations. After this, it was necessary to ramp up the speed of the blades as opposed to going from steady state to a rotation of 200,000 revolutions per minute, the design speed. In addition to this, the memory allocation factor in the simulation setup must be turned from 1.0 to 1.2 for each case. This is further explained in Appendix D.

4. Results

The run completed with maximum root mean square (RMS) residuals converging to a value of approximately 1.0e-6 with double precision computing. The primary monitored property is the pressure ratio, which was set as a monitor point before the run started. Figure 24 and Figure 25 are representations of the pressure ratio as a function of the timestep.

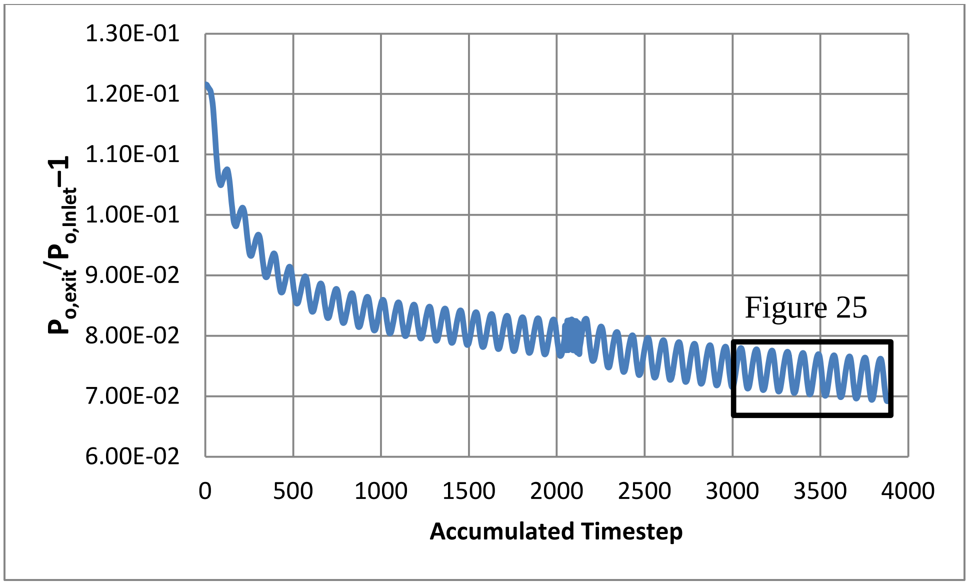

Figure 24. Pressure Ratio vs. Timestep for Two-Dimensional Run

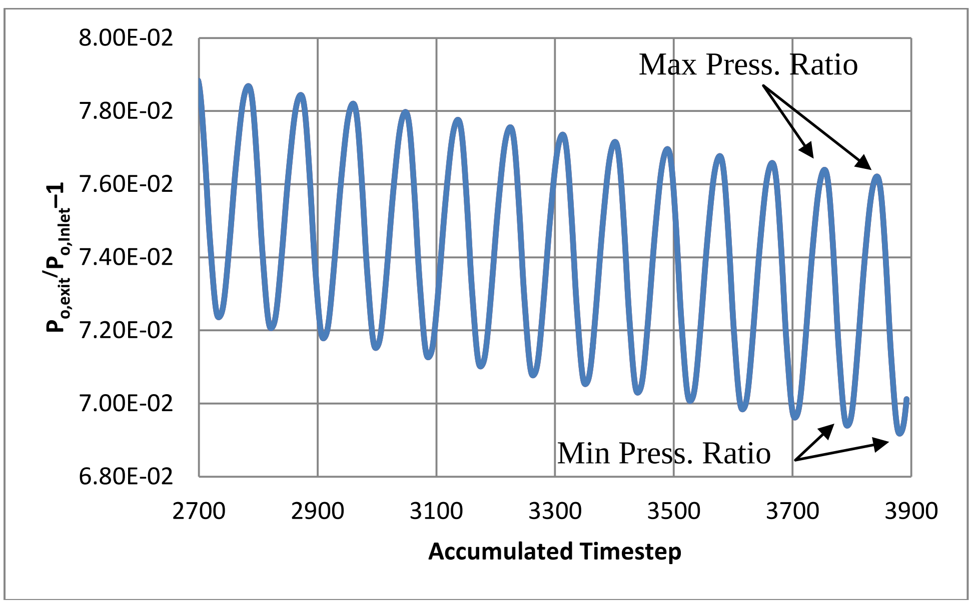

Figure 25. Pressure Ratio vs. Timestep for Two-Dimensional Run Zoomed In

As shown, the pressure ratio for this design is set to converge to an approximate value of 0.0735. This is more favorable than the first design, mainly because it produces a larger average magnitude of a pressure ratio. The two key points are the minimum and maximum values for this oscillating plot. The pressure distribution can be studied at the two points as shown in Figure 26 and Figure 27.

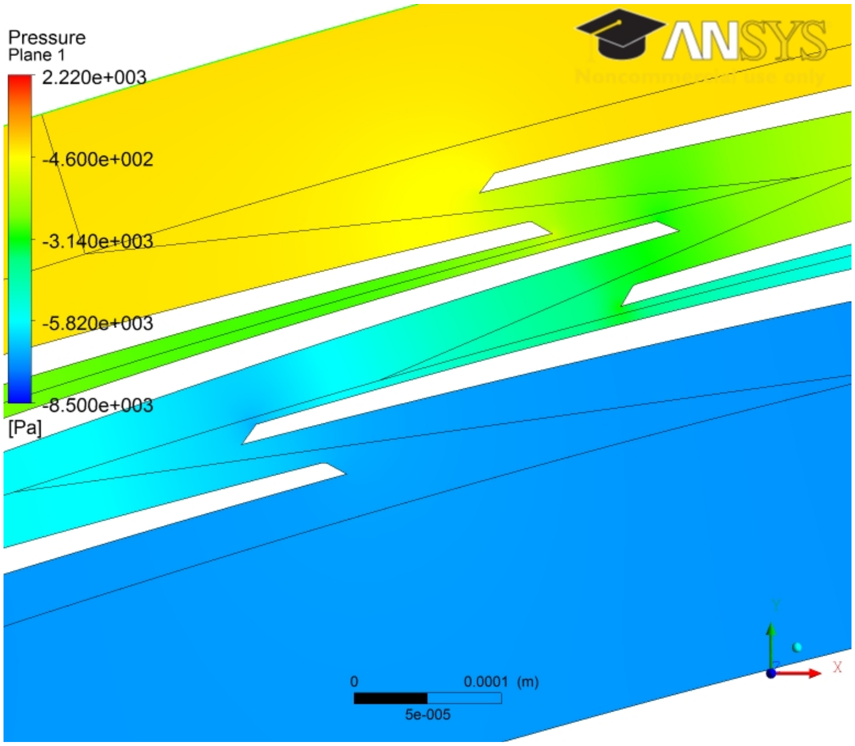

Figure 26.   Minimum Pressure Ratio Two-Dimensional 2nd Design

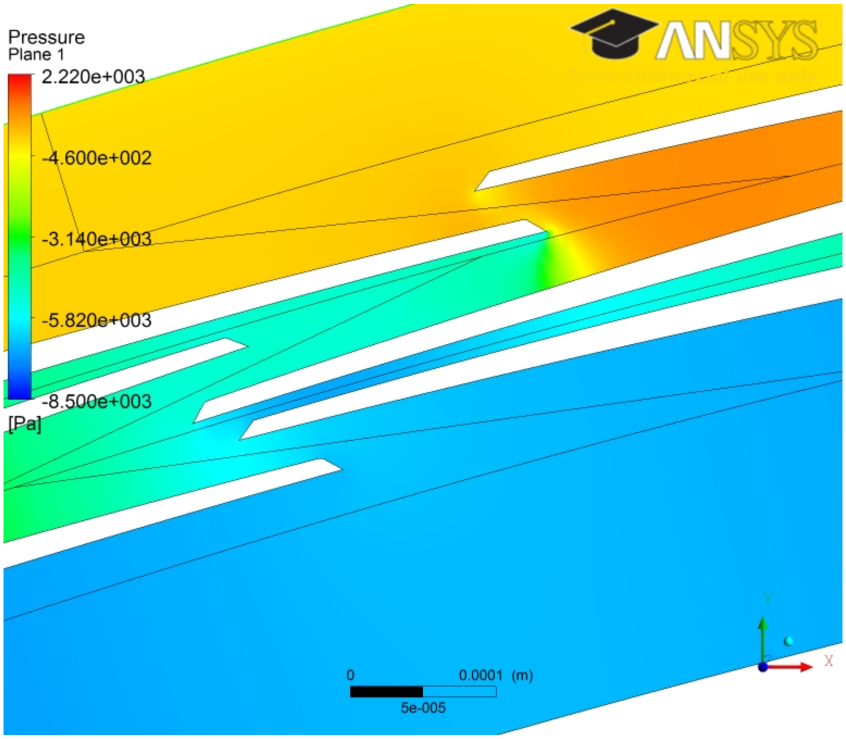

Figure 27. Maximum Pressure Ratio Two-Dimensional 2nd Design

The same plots can be compared with velocity vector distributions, as shown in Figure 28 and Figure 29.

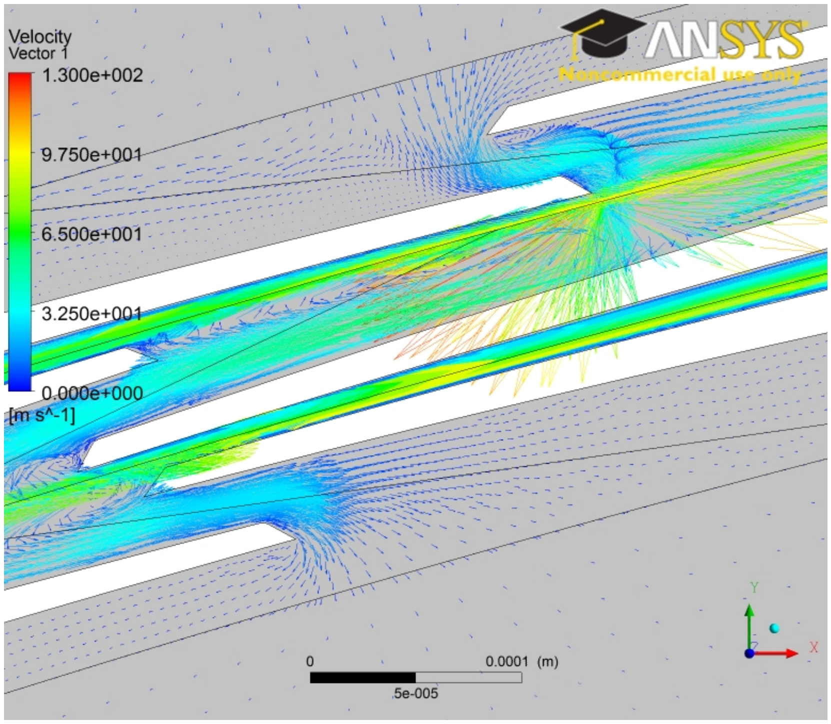

Figure 28. Velocity Distribution at Max Press Ratio, Two-Dimensional 2nd Design

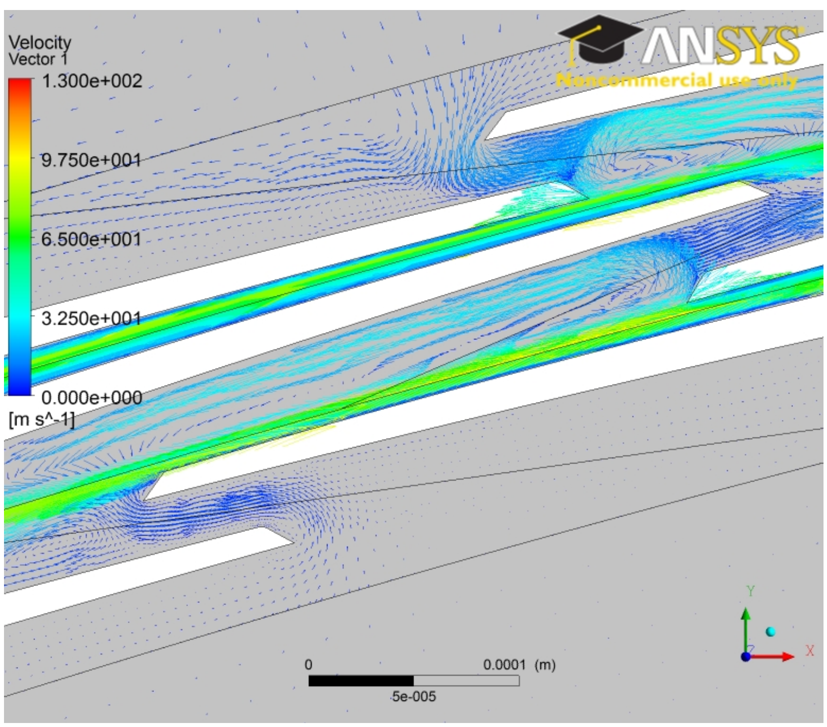

Figure 29. Velocity Distribution at Min Press Ratio, Two-Dimensional 2nd Design

As shown, at the minimum pressure ratios, there is a much larger amount of recirculation in between the blades leading to flow reversal and thus working against the pressure gradient the rotor is trying to build up across the stage. At the maximum pressure ratios, the velocity vector field is much more aligned with the stage and therefore more effective in driving air out of the vacuum. This is inherent to the flow of this type of machine as the net flow will eventually be zero. This can also be seen through the point of view of streamlines in the flow as shown in Figure 30 and Figure 31.

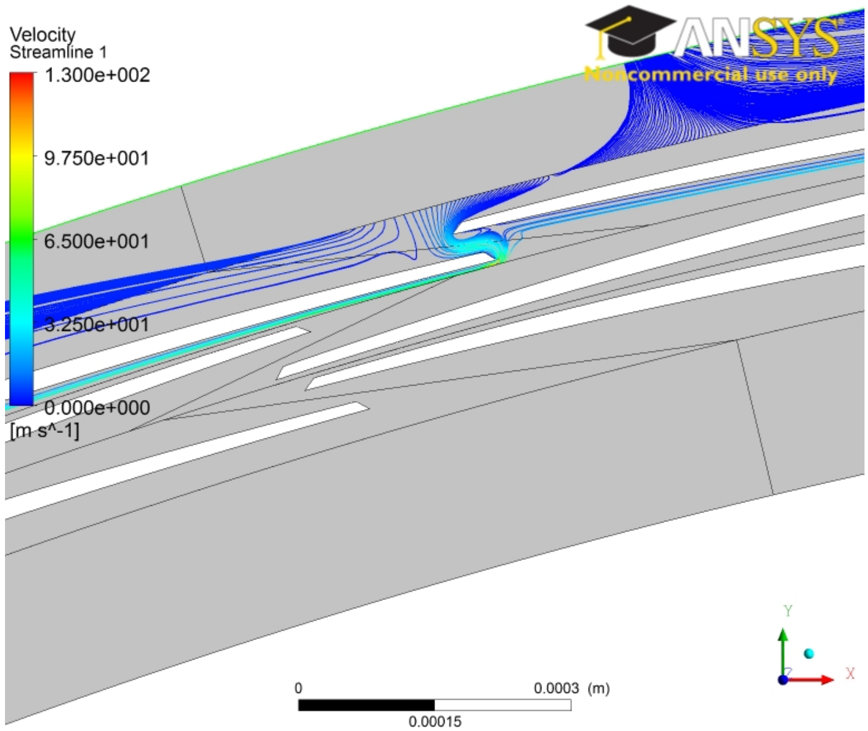

Figure 30. Minimum Pressure Ratio Streamlines

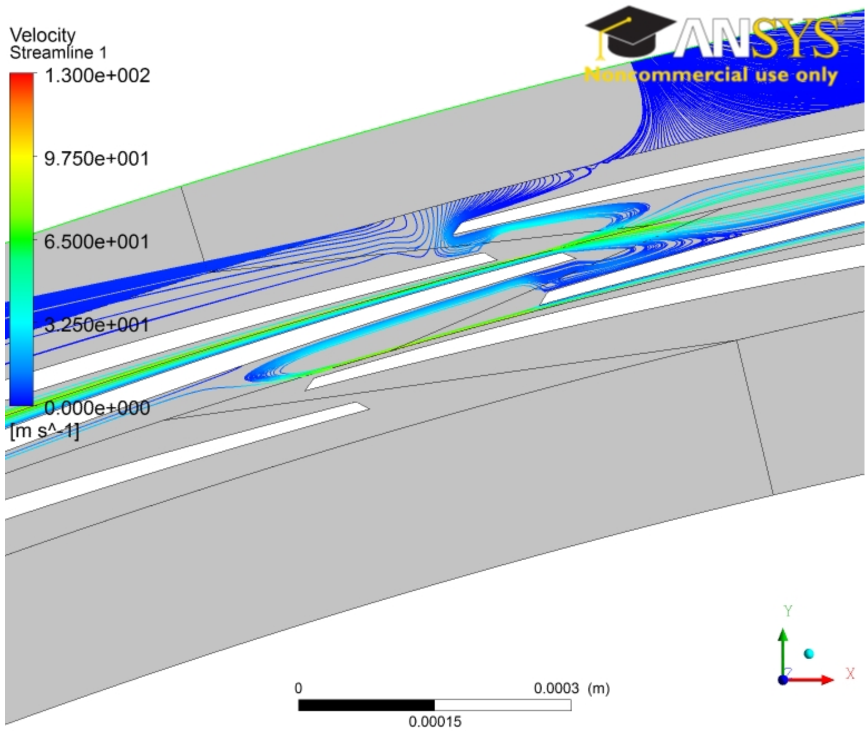

Figure 31. Maximum Pressure Ratio Streamline

In this case that is completely converged, the flow has been successfully driven out of the center of the pump, thereby creating a vacuum. Although some of the streamlines make it out of the pump, the recirculation is very apparent in these streamlines.

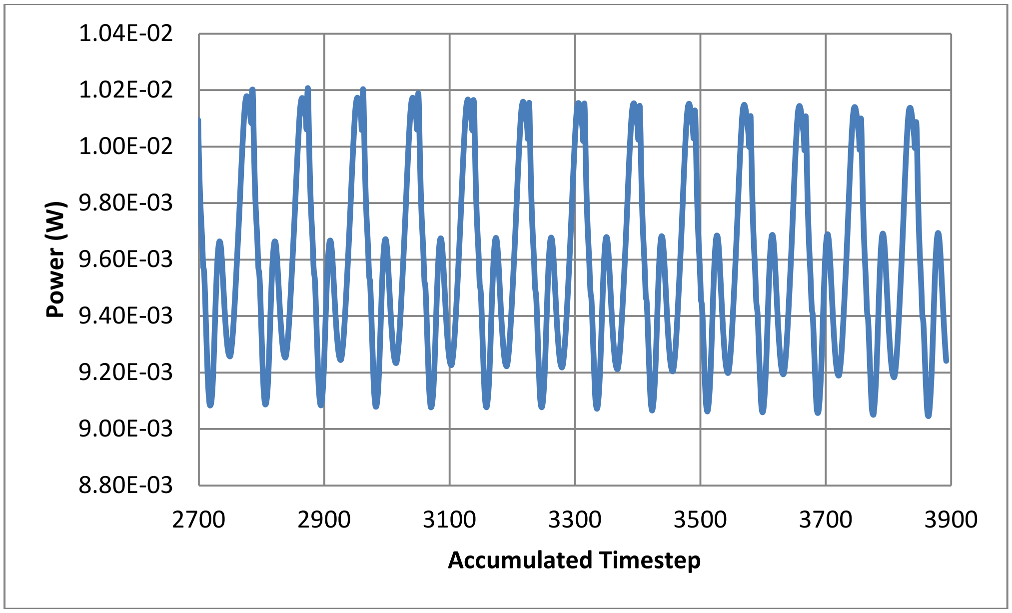

Figure 32. Single Blade Power for Two-Dimensional Design 2

The power converges to a mean value of approximately 9.6E-3 Watts. With 34 blades in this design, the power consumption of the entire blade row is 0.3264 watts. This value is smaller than that which will be seen in the three-dimensional case, namely because of the lack of the recirculatory region which will be seen in the three dimensional case over the tip gap.

THIS PAGE INTENTIONALLY LEFT BLANK

IV. THREE-DIMENSIONAL SIMULATIONS OF MICRO VACUUM PUMP

The three-dimensional simulation is necessary based on the tip gap region which is relatively larger than larger vacuum pumps. The three-dimensional modeling of the vacuum pump used the second iteration of the two-dimensional design but included a 5μm tip gap and the blade pedestals of the actual rotor geometry. The three-dimensional case required more elements because 5 elements were included in the tip gap region.

A. GEOMETRY

As mentioned, the three-dimensional geometry was based on the second iteration of two-dimensional geometry. Unlike the two-dimensional case, this consisted of an assembly of two different disks; one containing the rotor blades and the other containing the stator blades. Figure 33 shows the rotors blades and how they are etched on top of a pedestal which must also be modeled.

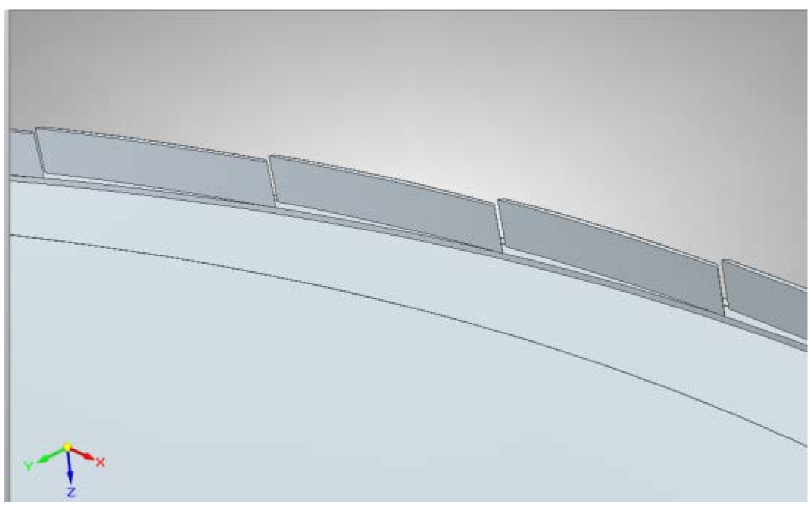

Figure 33. Rotor Blades for Three-Dimensional Simulation

The stator blades are very similar, the only difference being that there are two rows of blades to model the one and a half stage representation of the vacuum pump.

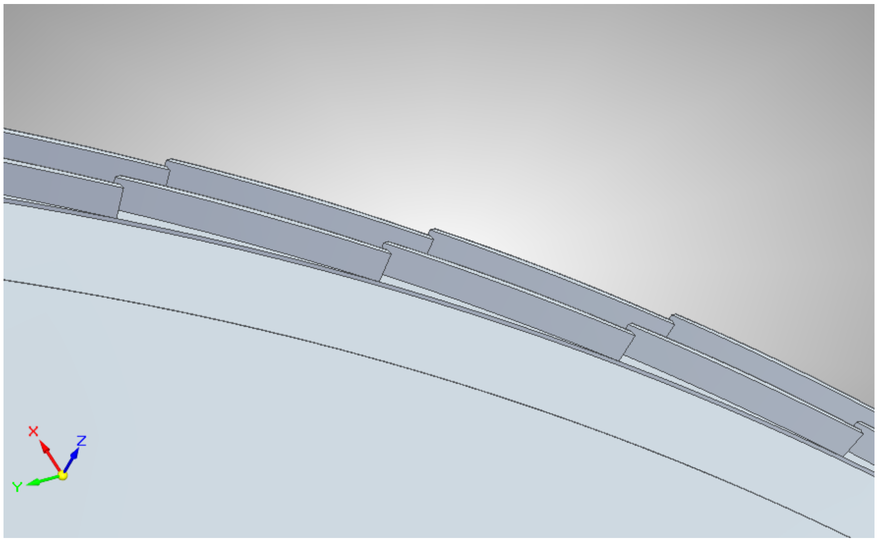

Figure 34. Stator Blades

These contain the same pedestals that the rotor contained. The two were then assembled, one on top of the other. A 5μm tip gap was left between the blade tip and the trench between the blade pedestals. After the design was modeled in SolidEdge, it was used as a negative to cut material out of an air volume thus ultimately modeling the air as opposed to the pump itself. After the geometry was finalized in SolidEdge, it was able to be imported into CFX for the setup of the fluid study. A graphical representation of the volumes containing the rotor and stator blades is shown in Figure 35.

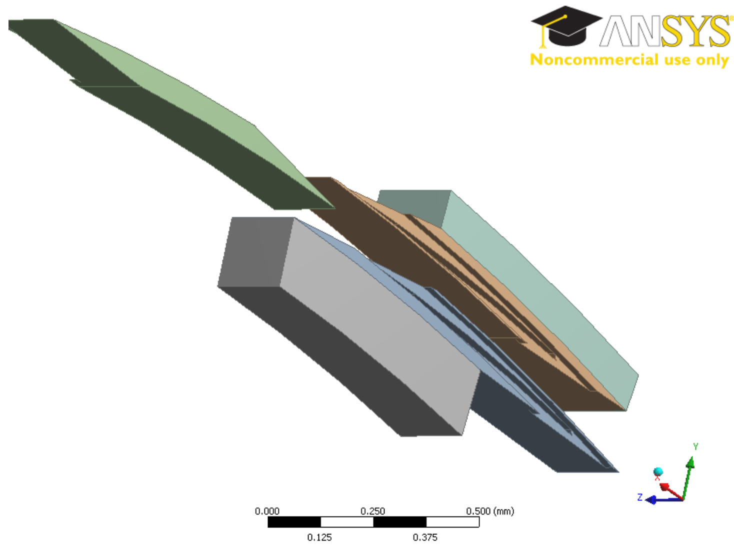

Figure 35. Computational Domain for Three-Dimensional Solver

B. MESHING

A crucial part of the three-dimensional modeling is that each control volume entirely contains the rotor and stator blades. If this is not done complications result with the symmetry condition between the two air molecules because one face has more elements than the other. This can be seen in the Figure 36 showing the fine mesh with just over 5 million elements.

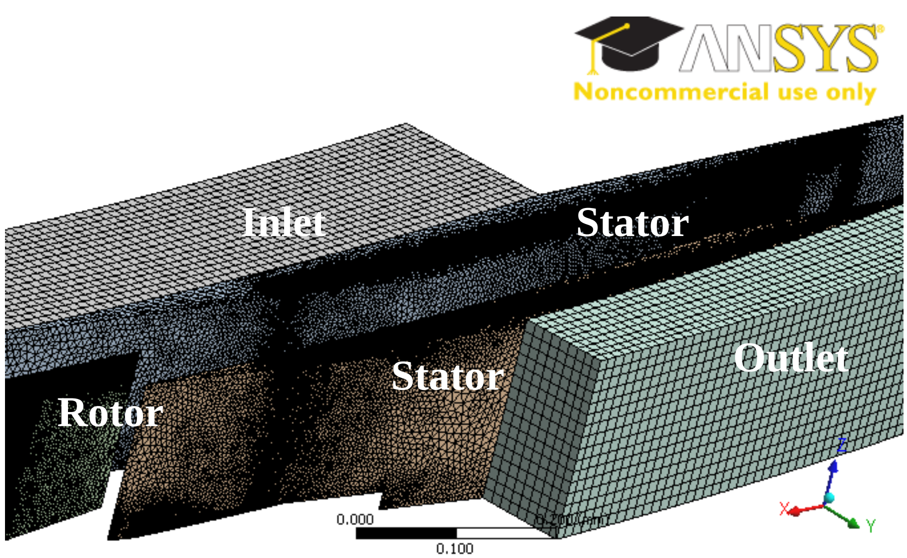

Figure 36. Mesh of Three-Dimensional Model

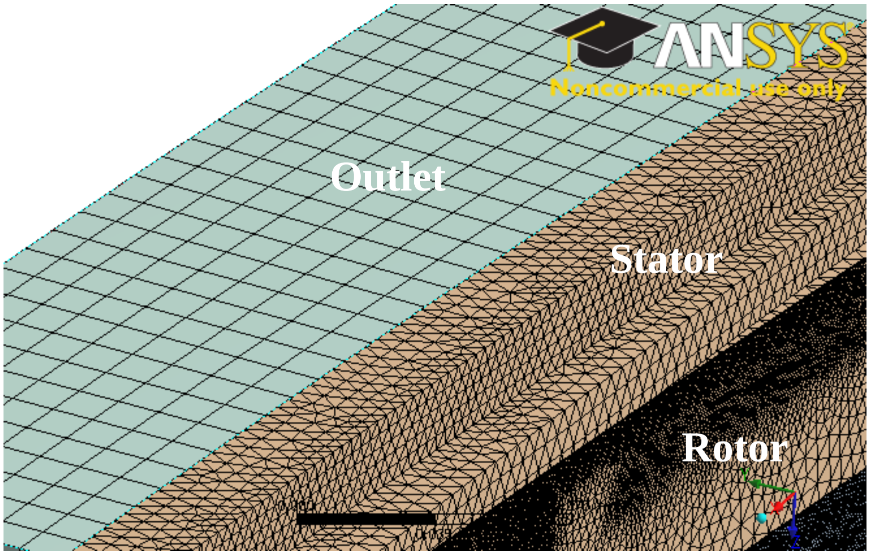

Figure 37. Mesh Interfaces of Three-DimensionalModel

As mentioned, the complexity of the three-dimensional mesh comes from the necessity to place a minimum of five nodes across any given gap, including the five micron tip gap. This also comes from the fact that span-wise there must be more than five elements across the blade like the two-dimensional case to accurately model three-dimensional effects.

C. SETUP

1. General Setup

Similar to the two-dimensional simulations, air as an ideal gas was used as the fluid at atmospheric conditions. The faces between the inlet and the first stator row, stator and rotor, etc were modeled as symmetry faces while the top and bottom faces were modeled as walls with a no slip condition. The assumption of laminar flow holds for this study because of the fact that the characteristic length and thus Reynolds's number is so small. More detailed setups can be found in Appendices F and G.

2. Steady State

To begin the steady state run, the stationary system was first allowed to run out for 200 iterations to provide initial results for the next step with a steady-state yet rotating assembly. However, rather than jumping from 0rpm to the specified 200,000rpm, it was necessary ramp up the speed gradually for 50-100 iterations at increasing speeds to prevent the solver from crashing. Ultimately, the 200,000rpm speed was run out to an RMS residual convergence of 1E-6.

2. Transient

Following the completion of the steady state run, the completed results were used as the input file to the transient run. The mixing model within the interfaces between the rotor and stator faces were changed to a Transient Rotor and the analysis was changed to a transient run with a total time of ten blade passing. A fixed timestep was used to avoid complications with the adaptive timestepping driving the timestep smaller and smaller, thus never reaching the total time assigned to this particular analysis type. A more detailed setup for this can be found in Appendix G.

D. RESULTS

In general, the three dimensional case, that included the five micron tip gaps produced a much less favorable pressure ratio than found in the two dimensional case. Although a tip gap of 5 microns may seem insignificant, for such a small scale pump, this is relatively large. Five microns on a 150 micron blade, makes up over 3% of the distance between the two rotating disks. There is an large tip leakage, as seen in Figure 38.

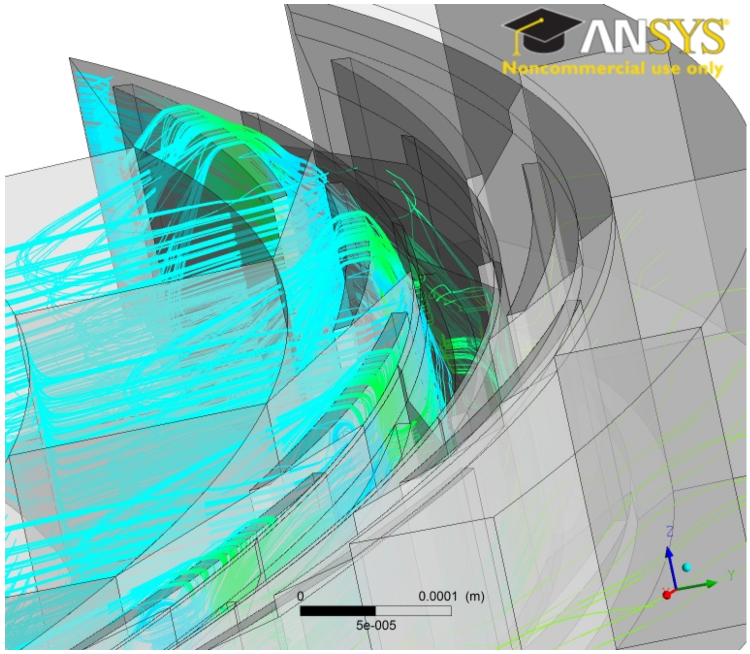

Figure 38. Tip Leakage over Stator Blade

Figure 38 shows streamlines originating from the inlet to the stage and a half turbopump section. The streamlines are seen to wrap around the stator blades expected but are then sucked back towards the center of the device over the tip gap. Similar phenomena are noticed when streamlines are studied in the tip gaps of the second stage of stator blades as well as over the rotor blades.

Another interesting aspect is the flow accelerating across the tip gap. As shown in Figure 39 the plane cutting the tip gap in half, actually shows regions of high velocity across the top of the blade.

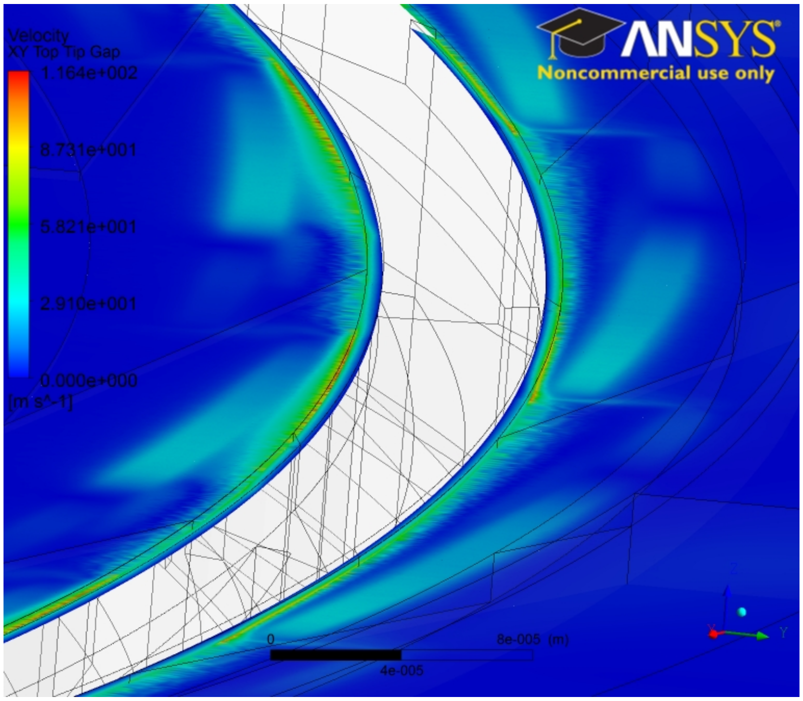

Figure 39. Velocity Distribution in Tip Gap

As mentioned, the pressure ratio was slightly not as desirable as the two-dimensional iteration, but Figure 40 and Figure 41 show both the full convergence of the pressure ratios in addition to a zoomed in shot of the last few blade passings where the solution was most converged, to RMS max residuals of about 1E-5. Recall that for the two-dimensional case the normalized converged pressure ratio was 7.22E-2.

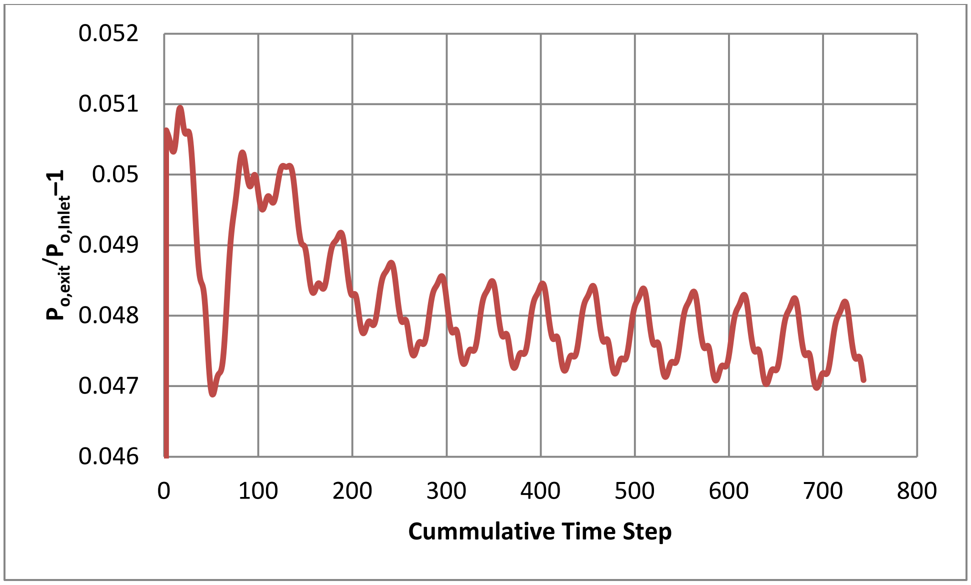

Figure 40. Pressure Ratio for Three-Dimensional Simulation

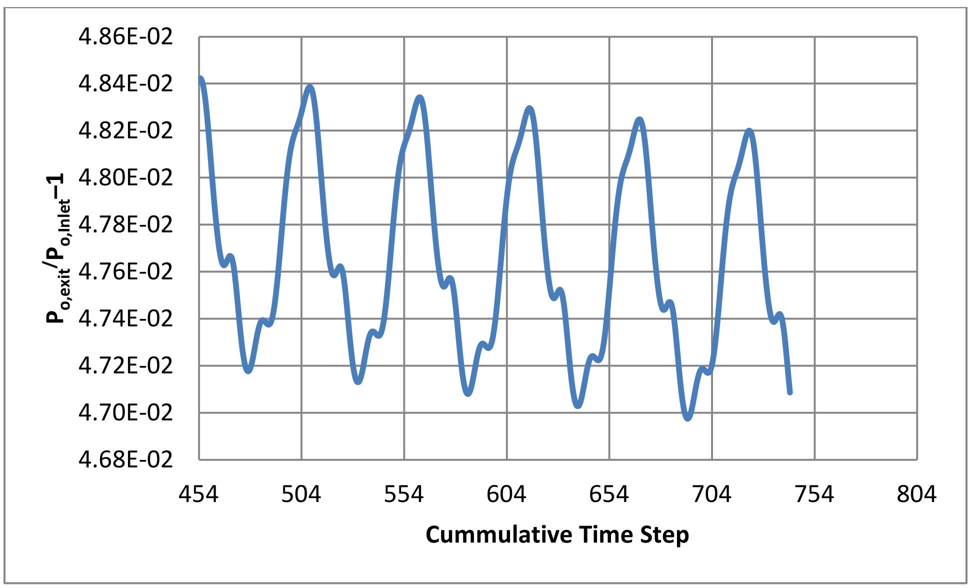

Figure 41. Zoomed Pressure Ratio for Three-Dimensional Simulation

The other output parameter of interest was the power consumed by the blade. The convergence history of this, showing the last iterations can be seen in Figure 42.

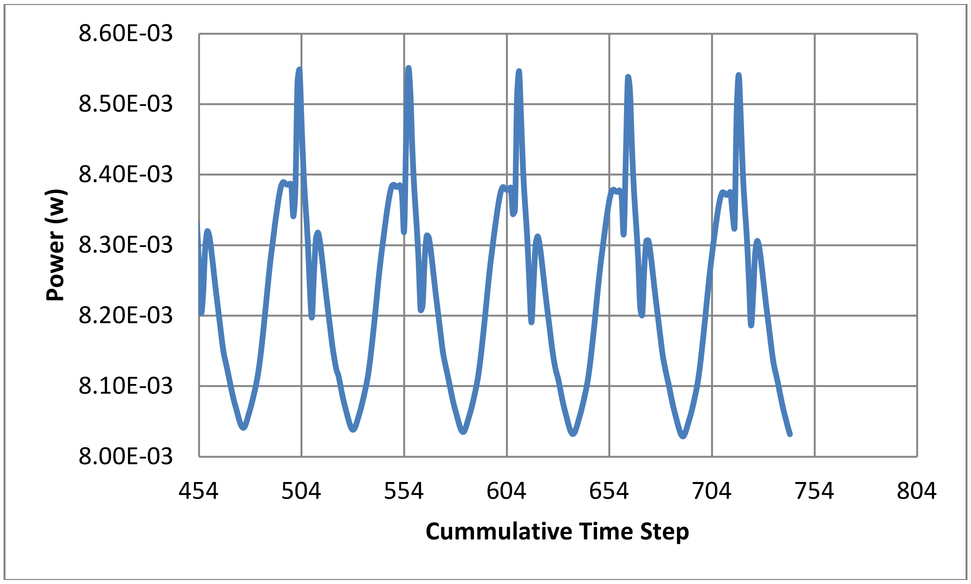

Figure 42. Single Blade Power for Three-Dimensional Simulation

This is a small power requirement, but this is due to the fact that it is only the power of one blade on a stage with 56 blades, with a potential for 100 stages on the pump. The power consumption for the entire stage will be about 0.4648 watts. The smaller stages will operate at lower pressure and thus consume less power. Studying the spikes in the pressure ratio and power plots, there are four key pointsin each blade passing- , as annotated in Figure 43, a minimum at timestep 692, a maximum at 722 and two other local maxima at 699 and 737.

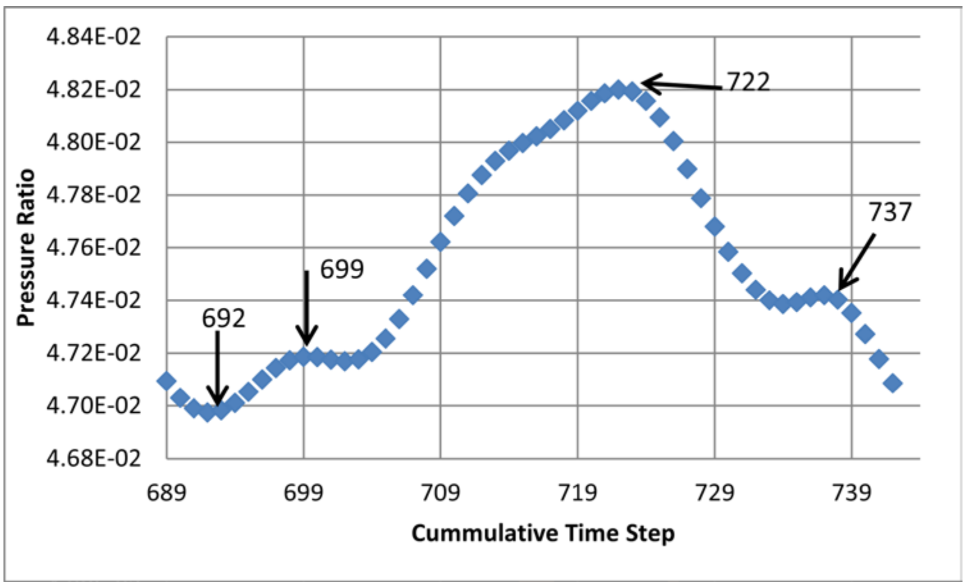

Figure 43. Critical Time Steps in Three-Dimensional Simulation

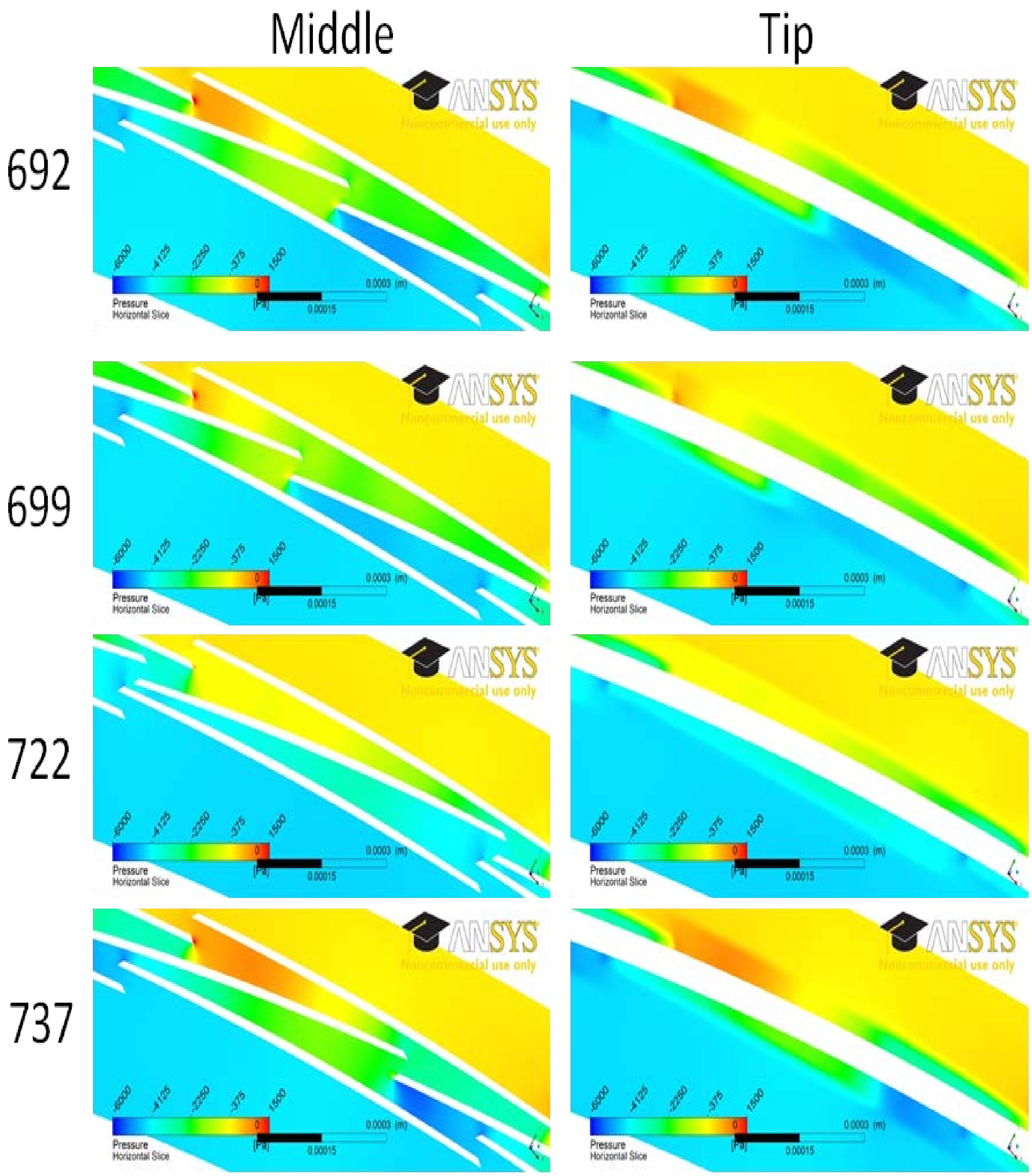

Figure 44. Pressure Plot Differences at Critical Time Steps

Figure 44 provides a comprehensive analysis as to why the various spikes in pressure and power occur over the course of a blade passing. It is very possible to see the stagnation point on the outer stator blade and the rotor as the rotor moves past. One of the driving factors in the pressure ratio across this three dimensional model is the fact that as the stagnation pressure increases at the leading edge of any of the blades, a high pressure region forms thereby forcing air through the tip gaps present. Thus, at the 692 time step, Figure 44 shows a minimum pressure ratio and Figure 44 show a large stagnation pressure at the end of the rotor blade, driving flow in inward even though the pump is trying to pump outward. Similarly, at timestep 722 has the smallest stagnation pressures therefore leading to the most effective position of the rotor blade and the greatest pressure ratio.

V. CONCLUSIONS

It has been demonstrated that a commercial code, in this case, CFX, is capable of computing slip flow. The computational fluid dynamics validation showed it is possible to easily model slip flow at the wall at Knudsen number flows between 0.001 to 0.1. To achieve greater accuracy, a second order analysis for this phenomenon would better simulate flow at higher Knudsen numbers.

A roughing pump stage was designed and simulated using a two dimensional simulation. The stage consisted of a stationary inlet and outlet surrounding a rotor blade row. To account for tip losses, a three dimensional simulation was conducted. With the methods developed, it is possible to predict pressure ratio and power consumption of this particular stage. The final predicted pressure ratio of a stage with tip clearance is 1.0722 with power consumption of 0.4648 watts.

THIS PAGE INTENTIONALLY LEFT BLANK

VI. RECOMMENDATIONS

Further study on design optimization and second-order slip flow should be conducted. Using a three-dimensional geometry and mesh, the blades should be modified to improve effectiveness forcing air out of the pump as opposed to the typical airfoil seen in macro-scale turbomachinery. The geometry and orientation of the blades should be designed in order to minimize tip leakage, most easily by minimize the large pressure developing on the inside of the stage, forcing air out in the wrong direction.

Once the general idea for a geometry is discovered, the setup of the CFD solver should be modified to utilize the slip flow expression vice the no slip condition that was used in this study. At the small scale and vacuum-like pressures, slip flow definitely occurs, especially through the tip leakages, thereby increasing the ability of air to escape across the tips, taking even more of a loss in pressure ratio. The full CFD analysis required to solve this problem is neither simple nor computationally inexpensive. It is necessary for a full three-dimensional solution incorporating a slip flow shear stress iterative expression, leading to a more accurate model of the MEMS-Scale Turbomachinery Based Vacuum Pump.

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF REFERENCES

[1] Advanced Test Concepts, "Helium Leak Detectors," http://www.atcinc.net/helium-leak-detectors.asp, 2012.

[2] Arkilic, E., Schmidt, M. and Breuer, K., "Gaseous Slip Flow in Long Microchannels," Journal of Microelectromechanical Systems, Vol.6, No. 2, pp. 167–178.

[3] Colin, S., "Gas Microflows in the Slip Flow Regime: A Review on Heat Transfer," ASME 2010 8th International Conference on Nanochannels, Microchannels, and Minichannels: Parts A and B. pp. 838–396.

[4] Cai, C., Sun, Q., and Boyd, I., "Gas flows in microchannels and microtubes," *Journal of Fluid Mechanics*, Vol. 589, pp. 305–314, August 2007.

[5] Knudsen, M., 1909, "Die Gesetze Der Molecular Stromung Und Die Inneren Reibungstromung Der Gase Durch Rohren," Annalen der Physik, Vol 28, pp. 75-130.

[6] Liou, W., Fang, Y., "Microfluid Mechanics," Blacklick, OH: McGraw-Hill Professional Publishing, 2005.

[7] Liu, J. and Tai, Y., "MEMS for Pressure Distribution Studies of Gaseous Flows in Microchannels," *8th Annual International Workshop MEMS. An Investigation of Micro Structures, Actuators, Machines, and Systems.* pp. 116–81.

[8] Mirshekarl, G. and Brouillette, M., "Compressible Microchannel Flow," *Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannel.* pp. 433–441.

[9] Pfahler, J., Harley, J., Bau, H., and Zemal, J., 1991, "Gas and Liquid Flow in Small Channels," Micromechanical Sensor, Actuators and Systems, ASME Winter Annual Meeting, pp. 49-59.

[10] Tison, S.A., "Experimental data and theoretical modelingg of gas flows through metal capillary leaks," *Vacuum*, Vol. 44, pp. 1175–1993, January 25, 1993.

[11] University of California, Berkley, "Description of a Basic Vacuum System," from Lab Manual for UC Berkley Microlab, Chapter 6.

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX A KNUDSEN NUMBER AT VARYING PRESSURES

No.	Bolt Cnst	Temp	Pres	Mol diam	Viscosity	L	Mn fr pth (2)	Knudsen
	k [m2.kg.s-2]	T [K]	P [Pa]	σ [m]	µ [N.s/m^2]	[m]	L	[-]
1	1.38E-23	288.15	200	3.70E-10	1.80E-05	0.00005	3.27E-05	6.54E-01
2	1.38E-23	288.15	300	3.70E-10	1.80E-05	0.00005	2.18E-05	4.36E-01
3	1.38E-23	288.15	400	3.70E-10	1.80E-05	0.00005	1.64E-05	3.27E-01
4	1.38E-23	288.15	500	3.70E-10	1.80E-05	0.00005	1.31E-05	2.62E-01
5	1.38E-23	288.15	600	3.70E-10	1.80E-05	0.00005	1.09E-05	2.18E-01
6	1.38E-23	288.15	700	3.70E-10	1.80E-05	0.00005	9.34E-06	1.87E-01
7	1.38E-23	288.15	800	3.70E-10	1.80E-05	0.00005	8.18E-06	1.64E-01
8	1.38E-23	288.15	900	3.70E-10	1.80E-05	0.00005	7.27E-06	1.45E-01
9	1.38E-23	288.15	1000	3.70E-10	1.80E-05	0.00005	6.54E-06	1.31E-01
10	1.38E-23	288.15	2000	3.70E-10	1.80E-05	0.00005	3.27E-06	6.54E-02

Table 1. Mean Free Path and Knudsen Number Calculations

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX B CFX SETUP FOR POISEULLE FLOW VALIDATION

Analysis Type	Basic Settings
	• External Solver Coupling
	  o Option: None
	• Analysis Type
	  o Option: **Steady State**

Default Domain Modified	Basic Settings
	• Location & Type
	  o Location: B30
	  o Domain Type: Fluid Domain
	  o Coordinate Frame: Coord 0
	• Fluid and Particle Definitions…
	  o Fluid 1
	    ▪ Option: Material Library
	    ▪ Material: Air Ideal Gas
	    ▪ Morphology
	      • Option: Continuous Fluid
	    ▪ Minimum Volume Fraction: unchecked
	• Domain Models
	  o Pressure
	    ▪ Reference Pressure: 200 [Pa]**
	    *** THIS CHANGED BASED ON DESIRED KNUDSEN NUMBER*
	  o Buoyancy Model
	    ▪ Option: Non Buoyant
	  o Domain Motion
	    ▪ Option: Stationary
	    ▪ Mesh Deformation: None

Default Domain Modified	Fluid Models
	• Heat Transfer
	  o Option: **Total Energy**
	  o Incl. Viscous Work Term: **CHECKED**
	• Turbulence
	  o Option: **None (Laminar)**
	• Combustion
	  o Option: None
	• Thermal Radiation
	  o Option: None
	• Electromagnetic Model: unchecked

Default Domain Modified	Initialization • Domain Initialization: unchecked
Default Domain Modified	Default Domain Modified Default | Basic Settings • Boundary Type: Symmetry • Location: F25.30,F27.30

Default Domain Modified	Inlet	Boundary Details • Mass And Momentum o Option: Interface Flux — Conservative • Turbulence o Option: Interface Flux — Conservative • Heat Transfer o Option : Interface Flux — Conservative
Default Domain Modified	Inlet	Basic Settings • Boundary Type: Opening • Location: Inlet • Coord Frame: unchecked
Default Domain Modified	Inlet	Boundary Details • Flow Regime Subsonic • Mass And Momentum o Option: Static Pres. and Dim o Relative Pressure: 0 [Pa] • Flow Direction o Option: Normal to Boundary Condition • Heat Transfer o Option : Opening Temperature o Opening Temperature: 288.15 [K]
Default Domain Modified	Inlet	Sources • Boundary Source: unchecked
Default Domain Modified	Outlet	Basic Settings • Boundary Type: Outlet • Location: Outlet • Coord Frame: unchecked
Default Domain Modified	Outlet	Boundary Details • Flow Regime Subsonic • Mass And Momentum o Option: Average Static Pressure o Relative Pressure: -2 [Pa] o Pres. Profile Blend 0.05 • Pressure Averaging o Option: Average Over Whole Outlet
Default Domain Modified	Outlet	Sources • Boundary Source: unchecked
Default Domain Modified	Symmetry	Basic Settings • Boundary Type: Symmetry • Location: Symmetry
Default Domain Modified	Knudsen Wall	Basic Settings • Boundary Type: Wall • Location: KnudsenWall

Default Domain Modified	Knudsen Wall (for Slip Flow Cases)	Boundary Details • Mass and Momentum o Option: Specified Shear • Shear Stress o Option: Cartesian Components o X Component: ShearSlip o Y Component: ShearSlip o Z Component: ShearSlip • Heat Transfer o Option: Adiabatic
Default Domain Modified	Knudsen Wall (for No Slip Cases)	Boundary Details • Mass and Momentum o Option: No Slip Wall o Wall Velocity: unchecked • Heat Transfer o Option: Adiabatic
Default Domain Modified	Knudsen Wall	Sources • Boundary Source: unchecked
Solver	No Category Settings	
Solver	Solution Units	Basic Settings • Mass Units: [kg] • Length Units: [m] • Time Units: [s] • Temperature Units [K] • Angle Units: CHECKED o Angle Units: [rad] • Solid Angle Units: CHECKED o Solid Angle Units: [sr]
Solver	Solver Control	Basic Settings • Advection Scheme o Option: **High Resolution** • Convergence Control o Min. Iterations **1** o Max. Iterations **10000** o Fluid Timescale Control ▪ Timescale Control: Auto Timescale ▪ Length Scale Option Conservat. ▪ Timescale Factor 1.0 ▪ Maximum Timescale unchecked • Convergence Criteria o Residual Type: **RMS** o Residual Target: **2e-22** o Conservation Target: unchecked • Elapsed Wall Clock Time Control: unchecked • Interrupt Control: unchecked
Solver	Solver Control	Equation Class Settings • Equation Class: Continuity • Continuity: unchecked

Solver	Solver Control	Advanced Options • Dynamic Model Control: CHECKED o Turbulence Control: unchecked o Hydro Control: unchecked • Pressure Level Information: unchecked • Body Forces: unchecked • Interpolation Scheme: unchecked • Temperature Damping: unchecked • Velocity Pressure Coupling: unchecked • Compressibility Control: unchecked
Solver	Output Control	Results • Option: Standard • File Compression: Default • Output Equation Residuals: unchecked • Extra Output Variable List unchecked
Solver	Output Control	Backup Results: Blank
Solver	Output Control	Monitor • Monitor Objects: unchecked
Expressions	MeanFreePath	boltzmann*ave(Total Temperature)@REGION:B30/(sqrt(2) *pi*ave(Absolute Pressure)@REGION:B30*MolecularDiam^2)
Expressions	MolecularDiam	3.7e-10[m]
Expressions	ShearNoSlip	ave(Wall Shear)@REGION:KnudsenWall
Expressions	ShearSlip	-ave(Velocity)@REGION:KnudsenWall*1.8E-5[N s m^-2]/(MeanFreePath)

APPENDIX C POISEULLE VALIDATION TABULATED RESULTS

		NO SLIP			SLIP		Plot	
Reference Pressure	Knudsen	Max Vel	Mean Vel	*2/3Max*	Max Vel	Mean Vel	LHS	Theory
200	6.54E-01	2.73037	1.82048	*1.8202467*	6.35379	5.44368	0.071244226	1.64E-01
300	4.36E-01	2.731	1.82089	*1.8206667*	5.1223	4.21176	0.058302185	1.09E-01
400	3.27E-01	2.73127	1.82106	*1.8208467*	4.50139	3.59062	0.049090895	8.18E-02
500	2.62E-01	2.73141	1.82116	*1.82094*	4.12396	3.21306	0.042134731	6.54E-02
600	2.18E-01	2.7315	1.82122	*1.821*	3.86795	2.95697	0.036645329	5.45E-02
700	1.87E-01	2.73156	1.82126	*1.82104*	3.68109	2.77004	0.032155427	4.67E-02
800	1.64E-01	2.7316	1.82129	*1.8210667*	3.5372	2.62611	0.02837569	4.09E-02
900	1.45E-01	2.73163	1.82131	*1.8210867*	3.24423	2.32674	0.018914506	3.63E-02

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX D SETUP FOR STEADY STATE 2D

Analysis Type	Basic Settings
	• External Solver Coupling
	    o Option: None
	• Analysis Type
	    o Option: **Steady State**

Rotor	Basic Settings
	• Location & Type
	    o Location: B272
	    o Domain Type: Fluid Domain
	    o Coordinate Frame: Coord 0
	• Fluid and Particle Definitions…
	    o Fluid 1
	        ▪ Option: Material Library
	        ▪ Material: Air Ideal Gas
	        ▪ Morphology
	            • Option: Continuous Fluid
	        ▪ Minimum Volume Fraction: unchecked
	• Domain Models
	    o Pressure
	        ▪ Reference Pressure: 1 [atm]
	    o Buoyancy Model
	        ▪ Option: Non Buoyant
	    o Domain Motion
	        ▪ Option: Rotating
	        ▪ Angular Velocity 200000 [rev min^-1]
	    o Axis Definition
	        ▪ Option Coordinate Axis
	        ▪ Rotation Axis Global Z
	        ▪ Mesh Deformation: None
	    o Mesh Deformation
	        ▪ Option None
	Fluid Models
	• Heat Transfer
	    o Option Total Energy
	    o Incl. Viscous Work Term Checked
	• Turbulence
	    o Option None (Laminar)
	• Combustion
	    o Option None
	• Thermal Radiation
	    o Option None
	• Electromagnetic Model Unchecked
	Initialization
	• Domain Initialization
	    o Frame Type Rotating
	    o Coord Frame Unchecked
	• Initial Conditions
	    o Velocity Type Cylindrical
	    o Cylindrical Velocity Components
	        ▪ Option Automatic

	o Velocity Scale: Unchecked • Static Pressure o Option: Automatic • Temperature o Option: Automatic	
Rotor	Rotor Default	Basic Settings • Boundary Type: Wall o Location: (automatically fills out) o Coord Frame: Unchecked o Frame Type: Rotating Boundary Details • Mass and Momentum o Option: No Slip Wall o Wall Velocity: Unchecked • Heat Transfer o Option: Adiabatic Sources • Boundary Source: Unchecked
Rotor	Rotor_Symmetry	Basic Settings • Boundary Type: Symmetry • Location: R1_Top, R1_Bottom
Stator	Basic Settings • Location & Type o Location: B1706,B2045 o Domain Type: Fluid Domain o Coordinate Frame: Coord 0 • Fluid and Particle Definitions… o Fluid 1 ▪ Option: Material Library ▪ Material: Air Ideal Gas ▪ Morphology • Option: Continuous Fluid ▪ Minimum Volume Fraction: unchecked • Domain Models o Pressure ▪ Reference Pressure: 1 [atm] o Buoyancy Model ▪ Option: Non Buoyant o Domain Motion ▪ Option: Stationary o Mesh Deformation ▪ Option: None Fluid Models • Heat Transfer o Option: Total Energy o Incl. Viscous Work Term: Checked • Turbulence o Option: None (Laminar) • Combustion o Option: None • Thermal Radiation	

<table>
<tr><td></td><td colspan="2">o Option None
• Electromagnetic Model Unchecked
Initialization
• Domain Initialization
o Frame Type Rotating
o Coord Frame Unchecked
• Initial Conditions
o Velocity Type Cylindrical
o Cylindrical Velocity Components
▪ Option Automatic
o Velocity Scale Unchecked
• Static Pressure
o Option Automatic
• Temperature
o Option Automatic</td></tr>
<tr><td>Stator</td><td>Stator Default</td><td>Basic Settings
• Boundary Type: Wall
o Location: (automatically fills out)
o Coord Frame Unchecked
o Frame Type Rotating
Boundary Details
• Mass and Momentum
o Option No Slip Wall
o Wall Velocity Unchecked
• Heat Transfer
o Option Adiabatic
Sources
• Boundary Source: Unchecked</td></tr>
<tr><td>Stator</td><td>Inlet</td><td>Basic Settings
• Boundary Type: Wall
• Location S1_Inlet
• Coord Frame: Unchecked
Boundary Details
• Mass and Momentum
o Option No Slip Wall
o Wall Velocity Unchecked
• Heat Transfer
o Option Adiabatic
Sources
• Boundary Source: Unchecked</td></tr>
<tr><td>Stator</td><td>Outlet</td><td>Basic Settings
• Boundary Type: Opening
• Location S2_Outlet
• Coord Frame: Unchecked
Boundary Details
• Flow Regime Subsonic
• Mass And Momentum
o Option: Entrainment
o Relative Pressure: 0 [Pa]
• Pressure Option Static Pressure
• Loss Coefficient Unchecked
• Heat Transfer</td></tr>
</table>

		o Option:	Opening Temp
		o Opening Temp	288.15 [K]
		Sources	
		• Boundary Source:	unchecked
Stator	S1_Symmetry	Basic Settings	
		• Boundary Type:	Symmetry
		• Location:	S1_Top, S1_Bottom
Stator	S2_Symmetry	Basic Settings	
		• Boundary Type:	Symmetry
		• Location:	S2_Top, S2_Bottom
Interfaces	R1_to_S2	Basic Settings	
		• Interface Type	Fluid Flow
		• Interface Side 1	
		o Domain	Rotor
		o Region List	R1_Outlet
		• Interface Side 2	
		o Domain	Stator
		o Region List	S2_Inlet
		• Interface Models	
		o Option	General Connection
		• Frame Change/ Mixing Model	
		o Option	Frozen Rotor
		o Rotational Offset	Unchecked
		• Pitch Change	
		o Automatic	
		Additional Interface Models	
		• Mass and Momentum	
		o Option	Conservative Interface Flux
		• Interface Model	
		o Option	None
		• Conditional Connection Contrl	Unchecked
		Mesh Connection	
		• Mesh Connection	
		o Option	GGI
		• Intersection Control	Unchecked
Interfaces	S1_to_R1	Basic Settings	
		• Interface Type	Fluid Flow
		• Interface Side 1	
		o Domain	Stator
		o Region List	S1_Outlet
		• Interface Side 2	
		o Domain	Rotor
		o Region List	R1_Inlet
		• Interface Models	
		o Option	General Connection
		• Frame Change/ Mixing Model	
		o Option	Frozen Rotor
		o Rotational Offset	Unchecked
		• Pitch Change	
		o Automatic	
		Additional Interface Models	
		• Mass and Momentum	

		o Option	Conservative Interface Flux
		• Interface Model	
		o Option	None
		• Conditional Connection Contrl	Unchecked
		Mesh Connection	
		• Mesh Connection	
		o Option	GGI
		• Intersection Control	Unchecked
Interfaces	SideSymmetry_R1	Basic Settings	
		• Interface Type	Fluid Flow
		• Interface Side 1	
		o Domain	Rotor
		o Region List	R1_Sym1
		• Interface Side 2	
		o Domain	Rotor
		o Region List	R1_Sym2
		• Interface Models	
		o Option	Rotational Periodicity
		• Axis Definition	
		o Option	Coordinate Axis
		o Rotational Axis	Global Z
		Mesh Connection	
		• Mesh Connection	
		o Option	Automatic
		• Intersection Control	Unchecked
Interfaces	SideSymmetry_S1	Basic Settings	
		• Interface Type	Fluid Flow
		• Interface Side 1	
		o Domain	Stator
		o Region List	S1_Sym1
		• Interface Side 2	
		o Domain	Stator
		o Region List	S1_Sym2
		• Interface Models	
		o Option	Rotational Periodicity
		• Axis Definition	
		o Option	Coordinate Axis
		o Rotational Axis	Global Z
		Mesh Connection	
		• Mesh Connection	
		o Option	Automatic
		• Intersection Control	Unchecked
Interfaces	SideSymmetry_S2	Basic Settings	
		• Interface Type	Fluid Flow
		• Interface Side 1	
		o Domain	Stator
		o Region List	S2_Sym1
		• Interface Side 2	
		o Domain	Stator
		o Region List	S2_Sym2

		• Interface Models
		  o Option — Rotational Periodicity
		• Axis Definition
		  o Option — Coordinate Axis
		  o Rotational Axis — Global Z
		Mesh Connection
		• Mesh Connection
		  o Option — Automatic
		• Intersection Control — Unchecked

Solver	Solution Units	Basic Settings
		• Mass Units: — [kg]
		• Length Units: — [m]
		• Time Units: — [s]
		• Temperature Units — [K]
		• Angle Units: — CHECKED
		  o Angle Units: — [rad]
		• Solid Angle Units: — CHECKED
		  o Solid Angle Units: — [sr]

Solver	Solver Control	Basic Settings
		• Advection Scheme
		  o Option: — High Resolution
		• Convergence Control
		  o Min. Iterations — 1
		  o Max. Iterations — 100
		  o Fluid Timescale Control
		    ▪ Timescale Control: — Auto Timescale
		    ▪ Length Scale Option — Conservat.
		    ▪ Timescale Factor — 1.0
		    ▪ Maximum Timescale — unchecked
		• Convergence Criteria
		  o Residual Type: — RMS
		  o Residual Target: — 1e-5
		  o Conservation Target: — unchecked
		• Elapsed Wall Clock Time Control: — unchecked
		• Interrupt Control: — unchecked
		Equation Class Settings
		• Equation Class: — Continuity, Energy Momentum
		• Continuity: — unchecked
		Advanced Options
		• Dynamic Model Control: — CHECKED
		  o Hydro Control: — unchecked
		• Pressure Level Information: — unchecked
		• Body Forces: — unchecked
		• Interpolation Scheme: — unchecked
		• Temperature Damping: — unchecked
		• Velocity Pressure Coupling: — unchecked
		Compressibility Control: — unchecked

Solver	Output Control	Results • Option: — Standard

		• File Compression: Default • Output Equation Residuals: unchecked • Extra Output Variable List unchecked Backup Results: Blank Monitor • Monitor Objects: Pressure Ratio* *Pressure Ratio is defined in expressions
Expressions	Absolute Omega	abs(200000 [rev min^-1])
Expressions	Exit MassFlow	massFlow()@REGION:S2_Outlet
Expressions	Inlet MassFlow	massFlow()@REGION:S1_Inlet
Expressions	Inlet Pressure	areaAve(Pressure)@S1_Inlet
Expressions	Power	(tBladeRow * Absolute Omega / 1 [rad])*(-1)
Expressions	Pressure Ratio	(101300 [Pa] /(101300 [Pa] +Inlet Pressure))-1
Expressions	tBladeRow	torque_z()@Rotor Default

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX E SETUP FOR TRANSIENT 2D

Analysis Type	Basic Settings • External Solver Coupling o Option: None • Analysis Type o Option: **Steady State**
Rotor	Basic Settings • Location & Type o Location: B272 o Domain Type: Fluid Domain o Coordinate Frame: Coord 0 • Fluid and Particle Definitions… o Fluid 1 ▪ Option: Material Library ▪ Material: Air Ideal Gas ▪ Morphology • Option: Continuous Fluid ▪ Minimum Volume Fraction: unchecked • Domain Models o Pressure ▪ Reference Pressure: 1 [atm] o Buoyancy Model ▪ Option: Non Buoyant o Domain Motion ▪ Option: Rotating ▪ Angular Velocity 200000 [rev min^-1] o Axis Definition ▪ Option Coordinate Axis ▪ Rotation Axis Global Z ▪ Mesh Deformation: None o Mesh Deformation ▪ Option None Fluid Models • Heat Transfer o Option Total Energy o Incl. Viscous Work Term Checked • Turbulence o Option None (Laminar) • Combustion o Option None • Thermal Radiation o Option None • Electromagnetic Model Unchecked Initialization • Domain Initialization o Frame Type Rotating o Coord Frame Unchecked • Initial Conditions o Velocity Type Cylindrical o Cylindrical Velocity Components ▪ Option Automatic

	o Velocity Scale: Unchecked • Static Pressure o Option: Automatic • Temperature o Option: Automatic	
Rotor	Rotor Default	Basic Settings • Boundary Type: Wall o Location: (automatically fills out) o Coord Frame: Unchecked o Frame Type: Rotating Boundary Details • Mass and Momentum o Option: No Slip Wall o Wall Velocity: Unchecked • Heat Transfer o Option: Adiabatic Sources • Boundary Source: Unchecked
Rotor	Rotor_Symmetry	Basic Settings • Boundary Type: Symmetry • Location: R1_Top, R1_Bottom
Stator	Basic Settings • Location & Type o Location: B1706,B2045 o Domain Type: Fluid Domain o Coordinate Frame: Coord 0 • Fluid and Particle Definitions… o Fluid 1 ▪ Option: Material Library ▪ Material: Air Ideal Gas ▪ Morphology • Option: Continuous Fluid ▪ Minimum Volume Fraction: unchecked • Domain Models o Pressure ▪ Reference Pressure: 1 [atm] o Buoyancy Model ▪ Option: Non Buoyant o Domain Motion ▪ Option: Stationary o Mesh Deformation ▪ Option: None Fluid Models • Heat Transfer o Option: Total Energy o Incl. Viscous Work Term: Checked • Turbulence o Option: None (Laminar) • Combustion o Option: None • Thermal Radiation	

	o Option: None • Electromagnetic Model: Unchecked Initialization • Domain Initialization o Frame Type: Rotating o Coord Frame: Unchecked • Initial Conditions o Velocity Type: Cylindrical o Cylindrical Velocity Components ▪ Option: Automatic o Velocity Scale: Unchecked • Static Pressure o Option: Automatic • Temperature o Option: Automatic	
Stator	Stator Default	Basic Settings • Boundary Type: Wall o Location: (automatically fills out) o Coord Frame: Unchecked o Frame Type: Rotating Boundary Details • Mass and Momentum o Option: No Slip Wall o Wall Velocity: Unchecked • Heat Transfer o Option: Adiabatic Sources • Boundary Source: Unchecked
Stator	Inlet	Basic Settings • Boundary Type: Wall • Location: S1_Inlet • Coord Frame: Unchecked Boundary Details • Mass and Momentum o Option: No Slip Wall o Wall Velocity: Unchecked • Heat Transfer o Option: Adiabatic Sources • Boundary Source: Unchecked
Stator	Outlet	Basic Settings • Boundary Type: Opening • Location: S2_Outlet • Coord Frame: Unchecked Boundary Details • Flow Regime: Subsonic • Mass And Momentum o Option: Entrainment o Relative Pressure: 0 [Pa] • Pressure Option: Static Pressure • Loss Coefficient: Unchecked • Heat Transfer

		o Option: Opening Temp o Opening Temp 288.15 [K] Sources • Boundary Source: unchecked
Stator	S1_Symmetry	Basic Settings • Boundary Type: Symmetry • Location: S1_Top, S1_Bottom
Stator	S2_Symmetry	Basic Settings • Boundary Type: Symmetry • Location: S2_Top, S2_Bottom
Interfaces	R1_to_S2	Basic Settings • Interface Type Fluid Flow • Interface Side 1 o Domain Rotor o Region List R1_Outlet • Interface Side 2 o Domain Stator o Region List S2_Inlet • Interface Models o Option General Connection • Frame Change/ Mixing Model o Option Frozen Rotor o Rotational Offset Unchecked • Pitch Change o Automatic Additional Interface Models • Mass and Momentum o Option Conservative Interface Flux • Interface Model o Option None • Conditional Connection Contrl Unchecked Mesh Connection • Mesh Connection o Option GGI • Intersection Control Unchecked
Interfaces	S1_to_R1	Basic Settings • Interface Type Fluid Flow • Interface Side 1 o Domain Stator o Region List S1_Outlet • Interface Side 2 o Domain Rotor o Region List R1_Inlet • Interface Models o Option General Connection • Frame Change/ Mixing Model o Option Frozen Rotor o Rotational Offset Unchecked • Pitch Change o Automatic Additional Interface Models • Mass and Momentum

		o Option	Conservative Interface Flux
		• Interface Model	
		o Option	None
		• Conditional Connection Contrl	Unchecked
		Mesh Connection	
		• Mesh Connection	
		o Option	GGI
		• Intersection Control	Unchecked
Interfaces	SideSymmetry_R1	Basic Settings	
		• Interface Type	Fluid Flow
		• Interface Side 1	
		o Domain	Rotor
		o Region List	R1_Sym1
		• Interface Side 2	
		o Domain	Rotor
		o Region List	R1_Sym2
		• Interface Models	
		o Option	Rotational Periodicity
		• Axis Definition	
		o Option	Coordinate Axis
		o Rotational Axis	Global Z
		Mesh Connection	
		• Mesh Connection	
		o Option	Automatic
		• Intersection Control	Unchecked
Interfaces	SideSymmetry_S1	Basic Settings	
		• Interface Type	Fluid Flow
		• Interface Side 1	
		o Domain	Stator
		o Region List	S1_Sym1
		• Interface Side 2	
		o Domain	Stator
		o Region List	S1_Sym2
		• Interface Models	
		o Option	Rotational Periodicity
		• Axis Definition	
		o Option	Coordinate Axis
		o Rotational Axis	Global Z
		Mesh Connection	
		• Mesh Connection	
		o Option	Automatic
		• Intersection Control	Unchecked
Interfaces	SideSymmetry_S2	Basic Settings	
		• Interface Type	Fluid Flow
		• Interface Side 1	
		o Domain	Stator
		o Region List	S2_Sym1
		• Interface Side 2	
		o Domain	Stator
		o Region List	S2_Sym2

		• Interface Models
		    o Option — Rotational Periodicity
		• Axis Definition
		    o Option — Coordinate Axis
		    o Rotational Axis — Global Z
		Mesh Connection
		• Mesh Connection
		    o Option — Automatic
		• Intersection Control — Unchecked

Solver	Solution Units	Basic Settings
		• Mass Units: [kg]
		• Length Units: [m]
		• Time Units: [s]
		• Temperature Units [K]
		• Angle Units: CHECKED
		    o Angle Units: [rad]
		• Solid Angle Units: CHECKED
		    o Solid Angle Units: [sr]

Solver	Solver Control	Basic Settings
		• Advection Scheme
		    o Option: High Resolution
		• Convergence Control
		    o Min. Iterations 1
		    o Max. Iterations 100
		    o Fluid Timescale Control
		        ▪ Timescale Control: Auto Timescale
		        ▪ Length Scale Option Conservat.
		        ▪ Timescale Factor 1.0
		        ▪ Maximum Timescale unchecked
		• Convergence Criteria
		    o Residual Type: RMS
		    o Residual Target: 1e-5
		    o Conservation Target: unchecked
		• Elapsed Wall Clock Time Control: unchecked
		• Interrupt Control: unchecked
		Equation Class Settings
		• Equation Class: Continuity, Energy Momentum
		• Continuity: unchecked
		Advanced Options
		• Dynamic Model Control: CHECKED
		    o Hydro Control: unchecked
		• Pressure Level Information: unchecked
		• Body Forces: unchecked
		• Interpolation Scheme: unchecked
		• Temperature Damping: unchecked
		• Velocity Pressure Coupling: unchecked
		Compressibility Control: unchecked

Solver	Output Control	Results • Option: Standard

		• File Compression: Default • Output Equation Residuals: unchecked • Extra Output Variable List unchecked Backup Results: Blank Monitor • Monitor Objects: Pressure Ratio* *Pressure Ratio is defined in expressions
Expressions	Absolute Omega	abs(200000 [rev min^-1])
Expressions	Exit MassFlow	massFlow()@REGION:S2_Outlet
Expressions	Inlet MassFlow	massFlow()@REGION:S1_Inlet
Expressions	Inlet Pressure	areaAve(Pressure)@S1_Inlet
Expressions	Power	(tBladeRow * Absolute Omega / 1 [rad])*(-1)
Expressions	Pressure Ratio	(101300 [Pa] /(101300 [Pa] +Inlet Pressure))-1
Expressions	tBladeRow	torque_z()@Rotor Default

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX F SETUP FOR STEADY STATE 3D

Analysis Type	Basic Settings
	• External Solver Coupling
	  o Option: None
	• Analysis Type
	  o Option: **Steady State**

Rotor	Basic Settings
	• Location & Type
	  o Location: B272
	  o Domain Type: Fluid Domain
	  o Coordinate Frame: Coord 0
	• Fluid and Particle Definitions…
	  o Fluid 1
	    ▪ Option: Material Library
	    ▪ Material: Air Ideal Gas
	    ▪ Morphology
	      • Option: Continuous Fluid
	    ▪ Minimum Volume Fraction: unchecked
	• Domain Models
	  o Pressure
	    ▪ Reference Pressure: 1 [atm]
	  o Buoyancy Model
	    ▪ Option: Non Buoyant
	  o Domain Motion
	    ▪ Option: Rotating
	    ▪ Angular Velocity 200000 [rev min^-1]
	  o Axis Definition
	    ▪ Option Coordinate Axis
	    ▪ Rotation Axis Global Z
	    ▪ Mesh Deformation: None
	  o Mesh Deformation
	    ▪ Option None
	Fluid Models
	• Heat Transfer
	  o Option Total Energy
	  o Incl. Viscous Work Term Checked
	• Turbulence
	  o Option None (Laminar)
	• Combustion
	  o Option None
	• Thermal Radiation
	  o Option None
	• Electromagnetic Model Unchecked
	Initialization
	• Domain Initialization
	  o Frame Type Rotating
	  o Coord Frame Unchecked
	• Initial Conditions
	  o Velocity Type Cylindrical
	  o Cylindrical Velocity Components
	    ▪ Option Automatic

	o Velocity Scale — Unchecked • Static Pressure o Option — Automatic • Temperature o Option — Automatic	
Rotor	Rotor Default	Basic Settings • Boundary Type: Wall o Location: (automatically fills out) o Coord Frame: Unchecked o Frame Type: Rotating Boundary Details • Mass and Momentum o Option: No Slip Wall o Wall Velocity: Unchecked • Heat Transfer o Option: Adiabatic Sources • Boundary Source: Unchecked
Stator	Basic Settings • Location & Type o Location: B1706,B2045 o Domain Type: Fluid Domain o Coordinate Frame: Coord 0 • Fluid and Particle Definitions… o Fluid 1 ▪ Option: Material Library ▪ Material: Air Ideal Gas ▪ Morphology • Option: Continuous Fluid ▪ Minimum Volume Fraction: unchecked • Domain Models o Pressure ▪ Reference Pressure: 1 [atm] o Buoyancy Model ▪ Option: Non Buoyant o Domain Motion ▪ Option: Stationary o Mesh Deformation ▪ Option: None Fluid Models • Heat Transfer o Option: Total Energy o Incl. Viscous Work Term: Checked • Turbulence o Option: None (Laminar) • Combustion o Option: None • Thermal Radiation o Option: None • Electromagnetic Model: Unchecked Initialization	

<table>
<tr><td></td><td colspan="2">• Domain Initialization
  o Frame Type: Rotating
  o Coord Frame: Unchecked
• Initial Conditions
  o Velocity Type: Cylindrical
  o Cylindrical Velocity Components
    ▪ Option: Automatic
  o Velocity Scale: Unchecked
• Static Pressure
  o Option: Automatic
• Temperature
  o Option: Automatic</td></tr>
<tr><td>Stator</td><td>Stator Default</td><td>Basic Settings
• Boundary Type: Wall
  o Location: (automatically fills out)
  o Coord Frame: Unchecked
  o Frame Type: Rotating
Boundary Details
• Mass and Momentum
  o Option: No Slip Wall
  o Wall Velocity: Unchecked
• Heat Transfer
  o Option: Adiabatic
Sources
• Boundary Source: Unchecked</td></tr>
<tr><td>Stator</td><td>Inlet</td><td>Basic Settings
• Boundary Type: Wall
• Location: S1_Inlet
• Coord Frame: Unchecked
Boundary Details
• Mass and Momentum
  o Option: No Slip Wall
  o Wall Velocity: Unchecked
• Heat Transfer
  o Option: Adiabatic
Sources
• Boundary Source: Unchecked</td></tr>
<tr><td>Stator</td><td>Outlet</td><td>Basic Settings
• Boundary Type: Opening
• Location: S2_Outlet
• Coord Frame: Unchecked
Boundary Details
• Flow Regime: Subsonic
• Mass And Momentum
  o Option: Entrainment
  o Relative Pressure: 0 [Pa]
• Pressure Option: Static Pressure
• Loss Coefficient: Unchecked
• Heat Transfer
  o Option: Opening Temp
  o Opening Temp: 288.15 [K]
Sources</td></tr>
</table>

		• Boundary Source:	unchecked
Interfaces	R1_to_S2	Basic Settings	
		• Interface Type	Fluid Flow
		• Interface Side 1	
		o Domain	Rotor
		o Region List	R1_Outlet
		• Interface Side 2	
		o Domain	Stator
		o Region List	S2_Inlet
		• Interface Models	
		o Option	General Connection
		• Frame Change/ Mixing Model	
		o Option	Frozen Rotor
		o Rotational Offset	Unchecked
		• Pitch Change	
		o Automatic	
		Additional Interface Models	
		• Mass and Momentum	
		o Option	Conservative Interface Flux
		• Interface Model	
		o Option	None
		• Conditional Connection Contrl	Unchecked
		Mesh Connection	
		• Mesh Connection	
		o Option	GGI
		• Intersection Control	Unchecked
Interfaces	S1_to_R1	Basic Settings	
		• Interface Type	Fluid Flow
		• Interface Side 1	
		o Domain	Stator
		o Region List	S1_Outlet
		• Interface Side 2	
		o Domain	Rotor
		o Region List	R1_Inlet
		• Interface Models	
		o Option	General Connection
		• Frame Change/ Mixing Model	
		o Option	Frozen Rotor
		o Rotational Offset	Unchecked
		• Pitch Change	
		o Automatic	
		Additional Interface Models	
		• Mass and Momentum	
		o Option	Conservative Interface Flux
		• Interface Model	
		o Option	None
		• Conditional Connection Contrl	Unchecked
		Mesh Connection	
		• Mesh Connection	
		o Option	GGI
		• Intersection Control	Unchecked

Interfaces	SideSymmetry_R1	Basic Settings
		• Interface Type — Fluid Flow
		• Interface Side 1
		  o Domain — Rotor
		  o Region List — R1_Sym1
		• Interface Side 2
		  o Domain — Rotor
		  o Region List — R1_Sym2
		• Interface Models
		  o Option — Rotational Periodicity
		• Axis Definition
		  o Option — Coordinate Axis
		  o Rotational Axis — Global Z
		Mesh Connection
		• Mesh Connection
		  o Option — Automatic
		• Intersection Control — Unchecked

Interfaces	SideSymmetry_S1	Basic Settings
		• Interface Type — Fluid Flow
		• Interface Side 1
		  o Domain — Stator
		  o Region List — S1_Sym1
		• Interface Side 2
		  o Domain — Stator
		  o Region List — S1_Sym2
		• Interface Models
		  o Option — Rotational Periodicity
		• Axis Definition
		  o Option — Coordinate Axis
		  o Rotational Axis — Global Z
		Mesh Connection
		• Mesh Connection
		  o Option — Automatic
		• Intersection Control — Unchecked

Interfaces	SideSymmetry_S2	Basic Settings
		• Interface Type — Fluid Flow
		• Interface Side 1
		  o Domain — Stator
		  o Region List — S2_Sym1
		• Interface Side 2
		  o Domain — Stator
		  o Region List — S2_Sym2
		• Interface Models
		  o Option — Rotational Periodicity
		• Axis Definition
		  o Option — Coordinate Axis
		  o Rotational Axis — Global Z
		Mesh Connection
		• Mesh Connection
		  o Option — Automatic

		• Intersection Control Unchecked
Solver	Solution Units	Basic Settings • Mass Units: [kg] • Length Units: [m] • Time Units: [s] • Temperature Units [K] • Angle Units: CHECKED o Angle Units: [rad] • Solid Angle Units: CHECKED o Solid Angle Units: [sr]
Solver	Solver Control	Basic Settings • Advection Scheme o Option: High Resolution • Convergence Control o Min. Iterations 1 o Max. Iterations 100 o Fluid Timescale Control ▪ Timescale Control: Auto Timescale ▪ Length Scale Option Conservat. ▪ Timescale Factor 1.0 ▪ Maximum Timescale unchecked • Convergence Criteria o Residual Type: RMS o Residual Target: 1e-5 o Conservation Target: unchecked • Elapsed Wall Clock Time Control: unchecked • Interrupt Control: unchecked Equation Class Settings • Equation Class: Continuity, Energy Momentum • Continuity: unchecked Advanced Options • Dynamic Model Control: CHECKED o Hydro Control: unchecked • Pressure Level Information: unchecked • Body Forces: unchecked • Interpolation Scheme: unchecked • Temperature Damping: unchecked • Velocity Pressure Coupling: unchecked Compressibility Control: unchecked
Solver	Output Control	Results • Option: Standard • File Compression: Default • Output Equation Residuals: unchecked • Extra Output Variable List unchecked Backup Results: Blank Monitor • Monitor Objects: Pressure Ratio* *Pressure Ratio is defined in expressions
Expressions	Absolute Omega	abs(200000 [rev min^-1])

Expressions	Exit MassFlow	massFlow()@REGION:S2_Outlet
Expressions	Inlet MassFlow	massFlow()@REGION:S1_Inlet
Expressions	Inlet Pressure	areaAve(Pressure)@S1_Inlet
Expressions	Power	(tBladeRow * Absolute Omega / 1 [rad])*(-1)
Expressions	Pressure Ratio	(101300 [Pa] /(101300 [Pa] +Inlet Pressure))-1
Expressions	tBladeRow	torque_z()@Rotor Default

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX G SETUP FOR TRANSIENT 3D

Analysis Type	Basic Settings
	• External Solver Coupling
	  o Option: None
	• Analysis Type
	  o Option: **Steady State**

Rotor	Basic Settings
	• Location & Type
	  o Location: B272
	  o Domain Type: Fluid Domain
	  o Coordinate Frame: Coord 0
	• Fluid and Particle Definitions…
	  o Fluid 1
	    ▪ Option: Material Library
	    ▪ Material: Air Ideal Gas
	    ▪ Morphology
	      • Option: Continuous Fluid
	    ▪ Minimum Volume Fraction: unchecked
	• Domain Models
	  o Pressure
	    ▪ Reference Pressure: 1 [atm]
	  o Buoyancy Model
	    ▪ Option: Non Buoyant
	  o Domain Motion
	    ▪ Option: Rotating
	    ▪ Angular Velocity 200000 [rev min^-1]
	  o Axis Definition
	    ▪ Option Coordinate Axis
	    ▪ Rotation Axis Global Z
	    ▪ Mesh Deformation: None
	  o Mesh Deformation
	    ▪ Option None
	Fluid Models
	• Heat Transfer
	  o Option Total Energy
	  o Incl. Viscous Work Term Checked
	• Turbulence
	  o Option None (Laminar)
	• Combustion
	  o Option None
	• Thermal Radiation
	  o Option None
	• Electromagnetic Model Unchecked
	Initialization
	• Domain Initialization
	  o Frame Type Rotating
	  o Coord Frame Unchecked
	• Initial Conditions
	  o Velocity Type Cylindrical
	  o Cylindrical Velocity Components
	    ▪ Option Automatic

	o Velocity Scale: Unchecked • Static Pressure o Option: Automatic • Temperature o Option: Automatic	
Rotor	Rotor Default	Basic Settings • Boundary Type: Wall o Location: (automatically fills out) o Coord Frame: Unchecked o Frame Type: Rotating Boundary Details • Mass and Momentum o Option: No Slip Wall o Wall Velocity: Unchecked • Heat Transfer o Option: Adiabatic Sources • Boundary Source: Unchecked
Stator	Basic Settings • Location & Type o Location: B1706,B2045 o Domain Type: Fluid Domain o Coordinate Frame: Coord 0 • Fluid and Particle Definitions… o Fluid 1 ▪ Option: Material Library ▪ Material: Air Ideal Gas ▪ Morphology • Option: Continuous Fluid ▪ Minimum Volume Fraction: unchecked • Domain Models o Pressure ▪ Reference Pressure: 1 [atm] o Buoyancy Model ▪ Option: Non Buoyant o Domain Motion ▪ Option: Stationary o Mesh Deformation ▪ Option: None Fluid Models • Heat Transfer o Option: Total Energy o Incl. Viscous Work Term: Checked • Turbulence o Option: None (Laminar) • Combustion o Option: None • Thermal Radiation o Option: None • Electromagnetic Model: Unchecked Initialization	

	• Domain Initialization
	    o Frame Type — Rotating
	    o Coord Frame — Unchecked
	• Initial Conditions
	    o Velocity Type — Cylindrical
	    o Cylindrical Velocity Components
	        ▪ Option — Automatic
	    o Velocity Scale — Unchecked
	• Static Pressure
	    o Option — Automatic
	• Temperature
	    o Option — Automatic	

Stator	Stator Default	Basic Settings
		• Boundary Type: — Wall
		    o Location: — (automatically fills out)
		    o Coord Frame — Unchecked
		    o Frame Type — Rotating
		Boundary Details
		• Mass and Momentum
		    o Option — No Slip Wall
		    o Wall Velocity — Unchecked
		• Heat Transfer
		    o Option — Adiabatic
		Sources
		• Boundary Source: — Unchecked

Stator	Inlet	Basic Settings
		• Boundary Type: — Wall
		• Location — S1_Inlet
		• Coord Frame: — Unchecked
		Boundary Details
		• Mass and Momentum
		    o Option — No Slip Wall
		    o Wall Velocity — Unchecked
		• Heat Transfer
		    o Option — Adiabatic
		Sources
		• Boundary Source: — Unchecked

Stator	Outlet	Basic Settings
		• Boundary Type: — Opening
		• Location — S2_Outlet
		• Coord Frame: — Unchecked
		Boundary Details
		• Flow Regime — Subsonic
		• Mass And Momentum
		    o Option: — Entrainment
		    o Relative Pressure: — 0 [Pa]
		• Pressure Option — Static Pressure
		• Loss Coefficient — Unchecked
		• Heat Transfer
		    o Option: — Opening Temp
		    o Opening Temp — 288.15 [K]
		Sources

		• Boundary Source:	unchecked
Interfaces	R1_to_S2	Basic Settings • Interface Type • Interface Side 1 o Domain o Region List • Interface Side 2 o Domain o Region List • Interface Models o Option • Frame Change/ Mixing Model o Option o Rotational Offset • Pitch Change o Automatic Additional Interface Models • Mass and Momentum o Option • Interface Model o Option • Conditional Connection Contrl Mesh Connection • Mesh Connection o Option • Intersection Control	 Fluid Flow Rotor R1_Outlet Stator S2_Inlet General Connection Frozen Rotor Unchecked Conservative Interface Flux None Unchecked GGI Unchecked
Interfaces	S1_to_R1	Basic Settings • Interface Type • Interface Side 1 o Domain o Region List • Interface Side 2 o Domain o Region List • Interface Models o Option • Frame Change/ Mixing Model o Option o Rotational Offset • Pitch Change o Automatic Additional Interface Models • Mass and Momentum o Option • Interface Model o Option • Conditional Connection Contrl Mesh Connection • Mesh Connection o Option • Intersection Control	 Fluid Flow Stator S1_Outlet Rotor R1_Inlet General Connection Frozen Rotor Unchecked Conservative Interface Flux None Unchecked GGI Unchecked

Interfaces	SideSymmetry_R1	Basic Settings • Interface Type — Fluid Flow • Interface Side 1 o Domain — Rotor o Region List — R1_Sym1 • Interface Side 2 o Domain — Rotor o Region List — R1_Sym2 • Interface Models o Option — Rotational Periodicity • Axis Definition o Option — Coordinate Axis o Rotational Axis — Global Z Mesh Connection • Mesh Connection o Option — Automatic • Intersection Control — Unchecked
Interfaces	SideSymmetry_S1	Basic Settings • Interface Type — Fluid Flow • Interface Side 1 o Domain — Stator o Region List — S1_Sym1 • Interface Side 2 o Domain — Stator o Region List — S1_Sym2 • Interface Models o Option — Rotational Periodicity • Axis Definition o Option — Coordinate Axis o Rotational Axis — Global Z Mesh Connection • Mesh Connection o Option — Automatic • Intersection Control — Unchecked
Interfaces	SideSymmetry_S2	Basic Settings • Interface Type — Fluid Flow • Interface Side 1 o Domain — Stator o Region List — S2_Sym1 • Interface Side 2 o Domain — Stator o Region List — S2_Sym2 • Interface Models o Option — Rotational Periodicity • Axis Definition o Option — Coordinate Axis o Rotational Axis — Global Z Mesh Connection • Mesh Connection o Option — Automatic

		• Intersection Control: Unchecked
Solver	Solution Units	Basic Settings • Mass Units: [kg] • Length Units: [m] • Time Units: [s] • Temperature Units [K] • Angle Units: CHECKED o Angle Units: [rad] • Solid Angle Units: CHECKED o Solid Angle Units: [sr]
Solver	Solver Control	Basic Settings • Advection Scheme o Option: High Resolution • Convergence Control o Min. Iterations 1 o Max. Iterations 100 o Fluid Timescale Control ▪ Timescale Control: Auto Timescale ▪ Length Scale Option Conservat. ▪ Timescale Factor 1.0 ▪ Maximum Timescale unchecked • Convergence Criteria o Residual Type: RMS o Residual Target: 1e-5 o Conservation Target: unchecked • Elapsed Wall Clock Time Control: unchecked • Interrupt Control: unchecked Equation Class Settings • Equation Class: Continuity, Energy Momentum • Continuity: unchecked Advanced Options • Dynamic Model Control: CHECKED o Hydro Control: unchecked • Pressure Level Information: unchecked • Body Forces: unchecked • Interpolation Scheme: unchecked • Temperature Damping: unchecked • Velocity Pressure Coupling: unchecked Compressibility Control: unchecked
Solver	Output Control	Results • Option: Standard • File Compression: Default • Output Equation Residuals: unchecked • Extra Output Variable List unchecked Backup Results: Blank Monitor • Monitor Objects: Pressure Ratio* *Pressure Ratio is defined in expressions
Expressions	Absolute Omega	abs(200000 [rev min^-1])

Expressions	Exit MassFlow	massFlow()@REGION:S2_Outlet
Expressions	Inlet MassFlow	massFlow()@REGION:S1_Inlet
Expressions	Inlet Pressure	areaAve(Pressure)@S1_Inlet
Expressions	Power	(tBladeRow * Absolute Omega / 1 [rad])*(-1)
Expressions	Pressure Ratio	(101300 [Pa] /(101300 [Pa] +Inlet Pressure))-1
Expressions	tBladeRow	torque_z()@Rotor Default

THIS PAGE INTENTIONALLY LEFT BLANK

INITIAL DISTRIBUTION LIST

1. Defense Technical Information Center
   Ft. Belvoir, Virginia

2. Dudley Knox Library
   Naval Postgraduate School
   Monterey, California

3. Dr. Knox T. Millsaps, Code ME
   Department of Mechanical and Aerospace Engineering
   Naval Postgraduate School
   Monterey, California

4. Dr. Garth V. Hobson, Code ME/HG
   Department of Mechanical and Aerospace Engineering
   Naval Postgraduate School
   Monterey, California

5. Dr. Anthony J. Gannon, Code ME/HG
   Department of Mechanical and Aerospace Engineering
   Naval Postgraduate School
   Monterey, California

6 Dr. Tayo Akinwande
   Program Manager
   DARPA
   Microsystems Technology Office (MTO)

7. Dr. Michael Wolfson
   Technical Monitor
   DARPA
   Microsystems Technology Office (MTO)